

Advanced Techniques used in Numerical Simulation for Deep-drawing Process

Valentin Oleksik^{1,}, Radu Breaz¹, Gabriel Racz¹, Paul Dan Brindasu², and Octavian Bologa¹*

¹Lucian Blaga University of Sibiu, Department of Machines and Equipment's, 10 Victoriei Bd, Sibiu, Romania

²Lucian Blaga University of Sibiu, Department of Industrial Engineering and Management, 10 Victoriei Bd, Sibiu, Romania

Abstract. The present paper analyse the main characteristics of the numerical simulation by finite element method of the deep-drawing processes. Also the authors' highlights the mathematical apparatus and the calculus method used for numerical simulations of metal forming processes in many of the current simulation software. The authors present the capabilities of the inverse analysis, direct analysis, implicit analysis (for springback simulation) and the optimisation analysis applied to explicit formulations.

1 Introduction

Metal forming technologies can be applied in a wide range of areas in the machines manufacturing industry, in fine mechanics, electro technics and electronics, the consumer goods industry etc. due to their technical-economic advantages: efficient usage of materials and energy, increase in productivity and manufacturing efficiency etc.

The current worldwide economic development is generated by a virtual explosion based on the implementation of sheet metal forming technologies instead of the classical machining technologies. The elaboration of new sheet metal forming technologies involves the usage of modern techniques for the computer-aided design of products and tools, simulating the forming process with the help of the finite elements method, adaptive process control, usage of numerical control machine-tools etc.

This has led to the obtaining of high-precision and high-quality parts with very complex shapes using a reduced number of operations.

All these cause the productivity to increase very much and the material losses to be minimal, so that the companies from countries with highly developed industry and scientific research occupy the first places in the production of such goods. This stage could be reached only as a consequence of the existence of well-organized fundamental and applicative researches and through the proper dissemination of the achieved results.

Any systematic research on the metal forming procedures is no longer possible without a mathematical description of the process. As of lately, in addition to the mathematical description by means of analytical procedures based on the theory of plasticity, the

* Corresponding author: valentin.oleksik@ulbsibiu.ro

modelling of forming processes using numerical methods gains more and more attention. Among these, the finite elements method is clearly standing out and, due to it being widely applied in engineering calculations, manages to achieve a very good matching of the calculated results with the experimentally determined ones. This has two main causes. On the one hand, the sustained succession of theoretical developments that materialize in new modelling software or facilities within consecrated software packages leads to the modelling of real phenomena using a very laborious mathematical apparatus. On the other hand, the appearance of powerful, relatively cheap computers is facilitating complex analyses and the large number of calculations specific for the finite elements method [1].

2 Considerations regarding the finite element method applied to the deep-drawing process

The analysis of metal forming using the finite elements method in the nonlinear domain presents a series of particularities that will be discussed in the following. Also, a short description of the mathematical apparatus and of the calculus method used for numerical simulations of metal forming processes will be made, indicating the manner in which this problem is tackled in most current calculation software packages.

In order to model the metal forming processes of this type, there need to be satisfied several conditions, such as: to observe the material balance laws from solid mechanics, to correctly interpret the material law, to correctly interpret the kinematics implied by the unfolding over time of the physical process, taking into account the contact phenomena and the local change of the material state. These conditions imply complications at the level of the mathematical apparatus for solving the equilibrium equations system. Thus, there will be needed [2], on the one hand, for the mathematical description, suitable parameters of the nodal displacements, adequate definitions of the stresses, adequate material laws, and on the other hand, for a suitable numerical treatment of the problems, it is desirable to have efficient formulations, stable and strong solving algorithms as well as a reliable and high capacity computation system. The equilibrium equation for a system with nonlinear behaviour is given in equation 1:

$$K(R, U) \cdot U = R \quad (1)$$

In this equation it can be seen that the system's stiffness matrix (K) depends on the load (R) and on the system's displacement (U) at the considered moment in time. The sources of this behaviour, in the case of metal forming, can be grouped [3] as follows:

- *Material nonlinearities*. They are caused by a nonlinear relationship between the applied stresses and the specific strains that occur in the deformed steel. This relationship cannot be defined precisely from a mathematical point of view, but it can be described by interpolating the experimental material data (yield curves).

- *Geometrical nonlinearities*. They are caused, on the one hand, by a nonlinear relationship between the displacements and the specific strains and on the other hand by the relationship applied forces - stresses. This type of nonlinearity is defined mathematically, but often it is difficult to treat mathematically. The mathematical separation of nonlinearities from the first two types is not unique. An example in this regard, for a plane element with four nodes, in two temporal phases of the analysis separated by a time increment Δt , is presented in figure 1, a. Here there can be noticed the nonlinear character of the dependence of specific strains on the nodal displacements and the fact that specific strains and rotations are no longer infinitesimal as is the case for loading in the elastic domain.

- *Nonlinearities of the boundary conditions.* The contact between the blank and the active elements and the inherent friction phenomena introduce these modifications of boundary conditions during the unfolding of an analysis. In the example from figure 1, b, as soon as the critical contact distance Δu decreases below a prescribed value, the nodal force changes abruptly, without any connection to the stress state existing in that element, simply due to the establishing of contact.

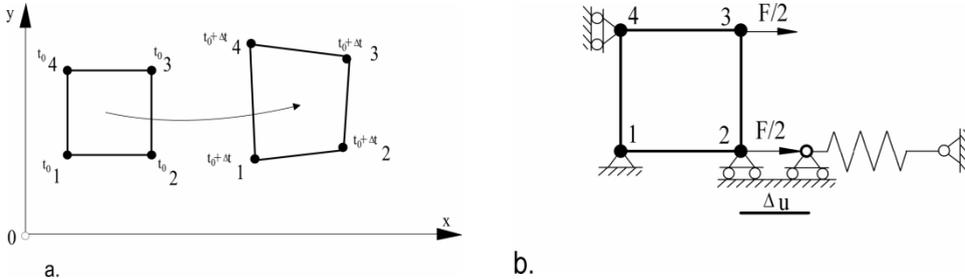


Fig. 1. Examples of nonlinearities a.) Example of a geometrical nonlinearity – plane element with four nodes in two temporal phases of the analysis. b.) Example of a boundary conditions nonlinearity – change in the nodal force due to the establishing of contact conditions

The finite elements analysis in the nonlinear domain is thus much more complex and more costly than the analysis in the linear domain. Also, the nonlinear analysis cannot be formulated as a set of linear equations and solved as such. Generally, solving these problems requires the applying of incremental solving schemes. This is done by correlating the unfolding of the forming process during time intervals Δt , called time increments. The solving schemes are iterative, often requiring the reprisal of the calculations over several cycles, in order to ensure the solution's convergence towards a static balance at the end of each time increment. Function of the tolerance imposed to the chosen convergence criterion, the analysis requires a larger or smaller number of iterations, for the same set of input data.

The calculation program used by the authors for modelling the cold metal forming process (Ls-Dyna), was selected due to its ability to describe and to take into account all three above-mentioned types of nonlinearities. The authors used an incremental formulation of the equilibrium equations (revised Lagrangian formulation) based on tracking the equilibrium state of the formable body at the time intervals $0, \Delta t, 2\Delta t, 3\Delta t, \dots, n\Delta t$ [4]. The calculations target all nodes of the finite elements lattice from the beginning of the forming process until its end. In the revised Lagrangian formulation, the stiffness matrix is assembled in the geometrical configuration of the current increment, while the results calculated at the end of the increment (stresses, specific strains, and nodal displacements) are associated to the revised nodal geometry in agreement with the calculated displacements. Among the available formulations, this one is the most suitable for modelling this forming process, characterized by a geometry that is continuously changing and by long distance displacements of some of the elements. In connection with this formulation there can be defined the material properties associated to the model, that have to characterize it under three aspects [5]:

- yield condition, that specifies the multiaxial stress state corresponding to the beginning of the plastic strains;
- flow law (normality law), that creates a relationship between the increase of the specific plastic deformation, the current stress state and the increase of stresses after yielding;
- hardening law, that indicate show the yielding condition changes during the plastic yielding.

The three aspects are summarized in figure 2. It shows the ellipses corresponding to the yield surface for the two-dimensional case in the Von Mises yield condition. The yield condition at the moment t is described by equation 2:

$${}^tF({}^t\sigma_{ij}, {}^t k) = 0 \tag{2}$$

where ${}^t k$ represents a status variable at the moment t , that depends on the natural deformation.

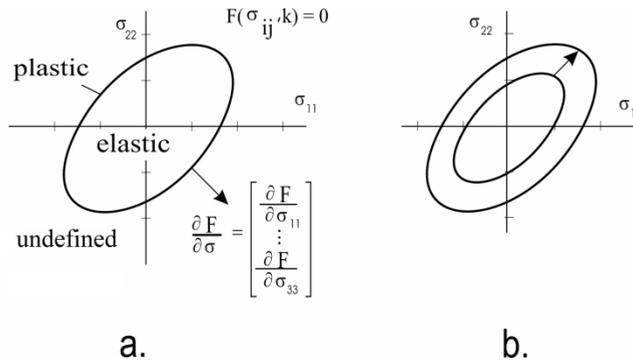


Fig. 2. Graphical representation of the material properties for the two-dimensional case

For the numerical analysis, the revised Lagrange formulation re-creates a reference framework at the beginning of each increment and requires as starting data the material’s real yield curve. These parameters related to the material’s plastic behavior are taken into account by the software and are processed in order to be incorporated into the stiffness matrix (K) within the equilibrium equation system (1). The implementation used is of elasto-plastic type, i.e. the elastic component of the specific strain is taken into account and the reaching of the plastic state is permanently tested by the calculus algorithm. In the following there will be briefly presented the calculus succession for the elements specific to the elasto-plastic formulation of the model. The algorithms used by the calculus software are stable and accurate for moderate increments of the specific strain. They are less accurate and have a weaker convergence for increases of the specific plastic deformation in an increment of more than ten times the corresponding specific elastic deformation. In this regard, when determining the time increment, there is an interval within whose limits it can be adjusted. The software calculates the relationship specific stresses-strains at the middle of the increment for each of the integration points based on a predicted specific incremental strain. For each first iterative cycle within a time increment, this prediction is based on the specific incremental strain from the previous time increment. For calculating the response stresses, there is used the average normal method, calculating a stiffness matrix at each increment. If the control variable at the end of each increment is within the limits of the imposed tolerance, no further reiterations are made. During reiteration, the values of the specific strains calculated at the previous iterations are considered to be predicted values for evaluating the stiffness matrix. In this manner, any negative influences that an uninspired initial prediction could have on the solution, are corrected.

For solving the equilibrium equation system, the numerical simulation software programs offer several incremental procedures. Among these, only two are considered for the analysis presented in the current paper: the Newton-Raphson method and the modified Newton-Raphson method. The Newton-Raphson method has following advantages [6]: it has a rapid (square) convergence as opposed to the modified Newton-Raphson method that has only a linear convergence, and it is possible to run analyses of problems with material

and geometric nonlinearities that stand out more. A major disadvantage of this method is the need to recalculate the stiffness matrix for each iteration, which can increase the computation time in the case of problems with a large number of elements in the grid. In this case, the Newton-Raphson has been preferred due to the medium size of the problem and the desire to reduce the number of iterations for a very high number of time increments.

3 Types of analyses solved with the finite elements method and their applications

Among the cold metal forming processes that can be simulated with the help of numerical methods, there can be mentioned: drawing, bending, calibrating, trimming, flanging, bulging, necking and extruding.

When simulating such a process there are two types of analyses that can be carried out: the inverse analysis and the direct analysis. Inverse analysis is an analysis that has as primary goal to determine the blank's shape (the part's developed surface) and to determine a presumed stress and strains state. The analysis starts from the final geometry of the part after the forming process, obtained by means of a CAD program that allows this and has as particularity the fact that neither the tools (die, punch, blankholder), nor the space between them or their kinematics are taken into account. The only data that need to be introduced are the part's geometry, its thickness and the material data. The possible obtained results are: the main strains, the variation of the material's thickness, the relative thickness reduction, the force and external work consumed in the process and the most important, the final shape of the blank. Figure 3 presents the desired meshed geometry, the initial blank shape dimensions, the formability diagram and the nodal displacement on Oz direction that represents, in fact, the punch displacement.

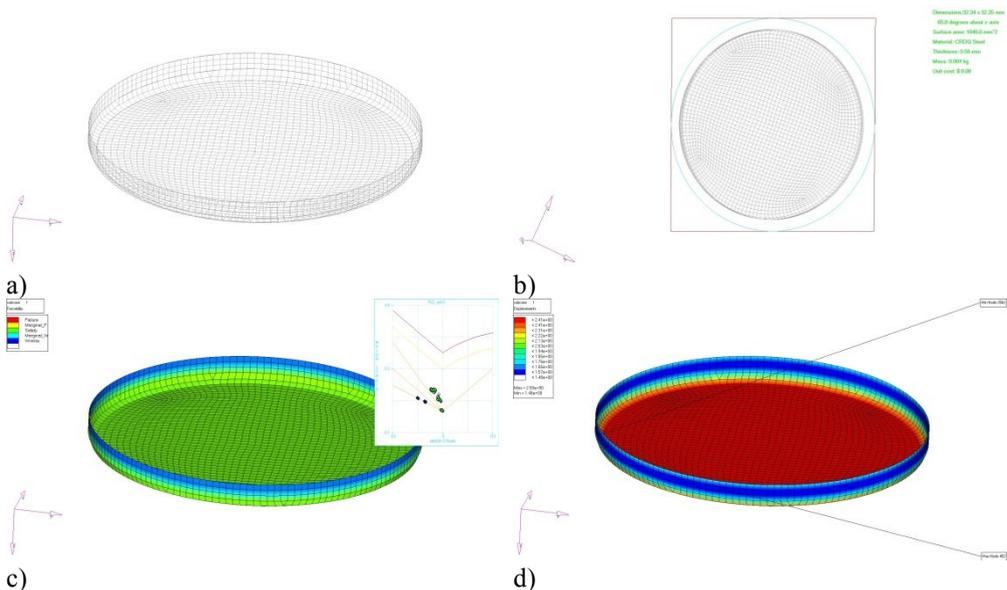


Fig. 3. The results obtained after the inverse analysis: a) The desired geometry; b) The blank shape obtained after the simulation; c) The formability diagram; d) The punch displacement

All the results can be displayed on both the final geometry and on the blank in order to emphasize problem areas. Also this type of analysis can be applied to determine the shape of a punching before the deep-drawing operation for a rectangular part with unequal walls.

These allow the making of the both operations (punching and blanking) on the same stamping die, before the deep-drawing process (fig. 4). The disadvantages of this type of analysis are represented by the fact that the tools and clearances are not taken into account.

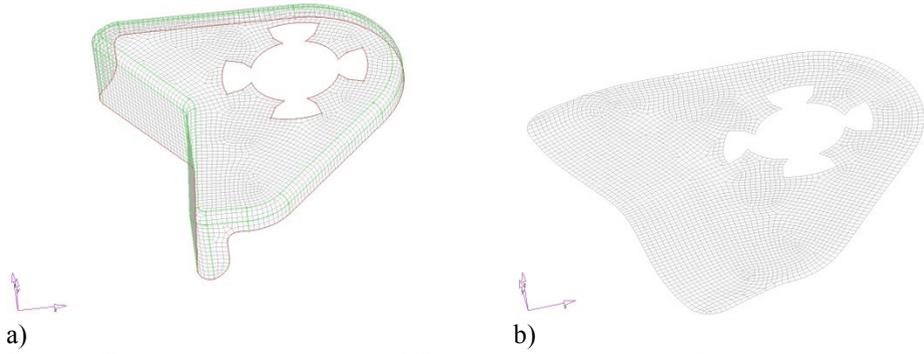


Fig. 4. a) The initial meshed geometry; b) The shape of the blank and of the punched hole

In order to eliminate these disadvantages, the numerical simulation software packages include also direct analyses (fig. 5) that start from the blank's shape and the final shape of the part is obtained after the simulation process.

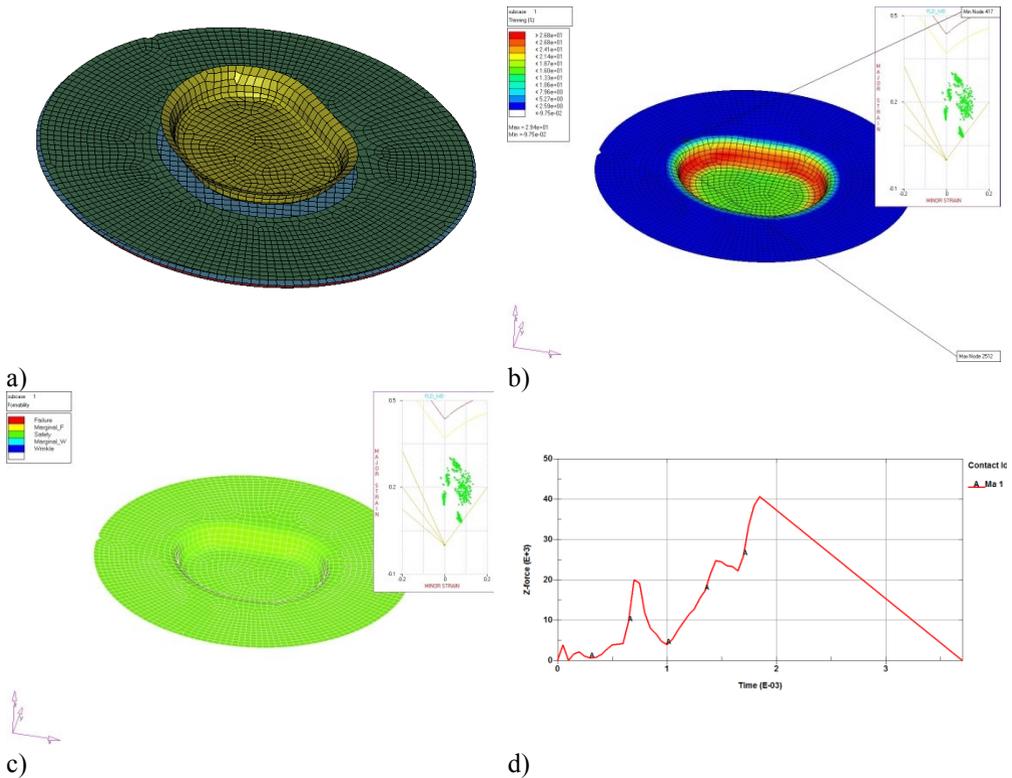
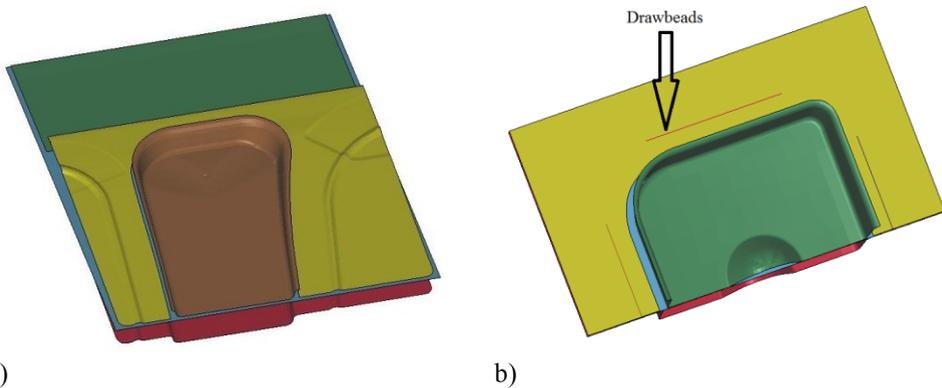


Fig. 5. The results obtained after the direct analysis: a) The meshed tools; b) The thinning; c) The formability diagram; d) The force variation

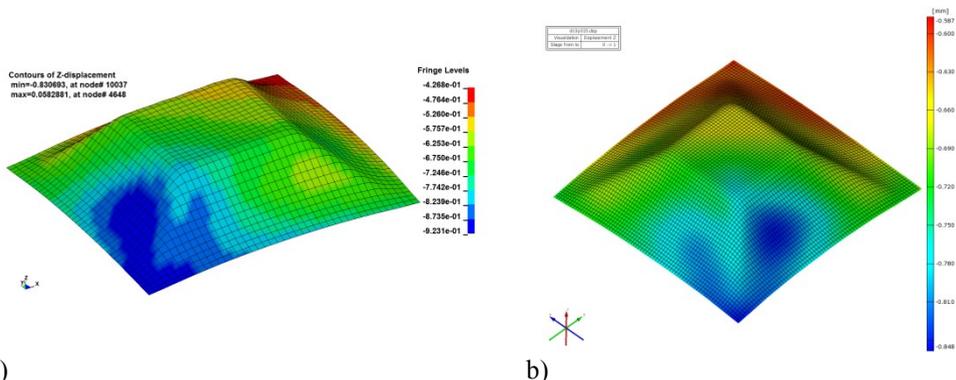
The data resulted from the analysis are mainly the same, but they are obtained with a smaller error percentage due to the simulation resembling more closely the real forming conditions, namely the inclusion of the tools in the simulation, as well as the space between punch and pressure plate, the kinematics of the punch (or of the die, if necessary), the force diving the blankholder element or the drawbeads. For these types of analyses there can be obtained also the formability diagrams that are graphs for all finite elements of the grid, in coordinates: major strain (ϵ_1) – minor strain (ϵ_2) (fig. 5, c). Function of the position on these graphs of the lines and curves specific for the processed material, there can be obtained data regarding the possibility of the occurrence of cracks or even of the fracture, or regarding excessive material thickening in various areas. Another important result is the one related to the variation of force during the process (fig. 5, d) that allows selecting the press for parts with complex geometry for which the force cannot be determined analytically.

Figure 6 presents two kinds of drawbeads, a real, geometric representation, that implies CAD modelling of each drawbead (Fig 6,a) and analytical drawbead which implies only the sketching of the drawbead and introducing the radius and the height (Fig 6, b).



a) b)
Fig. 6. Two different kinds of drawbeads used for the simulation of the deep-drawing process a) Real (geometric) drawbeads; b) Analytical (sketched) drawbeads

An important result facilitated by direct analysis is the obtaining of the elastic springback of the part's material after removing it from the die. This is a major advantage because elastic springback is an unwanted phenomenon of the modification of the shape and dimensions of a part subjected to a cold forming process and knowing the size of the elastic springback is essential for designing the forming die (fig. 7).



a) b)
Fig. 7. Springback variation a) Obtained by simulation after an implicit analysis; b) Measured on the part with an optical extensometer

Figure 7 presents a comparison between the results obtained for the springback after an implicit analysis performed after the direct analysis and the real, measured results obtained after the measured process done by Gom optical extensometer. The implicit analysis implies the “freezing” of the strain and stress state after the deep-drawing process.

4 Conclusions

The main conclusions of the paper are linked to the advantages of the simulations of the deep-drawing process: reducing the time spent for designing a die for deep-drawing, the possibility of calculating the process forces, the external work, cinematic energy, the possibility of determining the shape and the dimensions of the blank shape (based on the constant volume principle not on the unfolded area of the part), determining of the main strains, thickness reduction of the sheet and the possibility of comparing to the same data with the measured ones and the possibility to obtained the springback before the manufacturing of the die.

This research was supported by the project “Dezvoltarea departamentului de cercetare al societății COMPA S.A și obținerea unor rezultate inovatoare în domeniul industriei auto” (Acronim COMCD), code SMIS 2014+: 121359, funding grant nr 1/AXA1/1.1.1.A/18.05.2018, ID: P_34_469.

References

1. P. Kaldunski, J. Theor. App. Mech-Pol., **56**, 3 781-792 (2018)
2. M. A. Crisfield, J. Shri, K. L. Li, The Stress Analysis Group of the Institute of Physics, Sheffield, 3-13 (1993)
3. K.J. Bathe, M.D. Snyder, A.P. Cimento, IEE Conference Publication 2, **68**, 1-23 (2017)
4. K. Essa, P. Hartley J.R.T. Int. J. Mater. Form. **4**(4), 401-412 (2011)
5. P. Hartley, I. Pillinger, *Numerical Modelling of Material Deformation Processes. R,D & Applications*, Springer Verlag, (1992)
6. J.R.T. Huges, *The Finite Element Method Linear Static and Dynamic Finite Element Analysis*. Prentice-Hall International Edition, USA, (1987)