TECHNICAL UNIVERSITY OF CLUJ-NAPOCA

ACTA TECHNICA NAPOCENSIS

Series: Applied Mathematics and Mechanics
Vol. 55, Issue IV, 2012

NUMERICAL SIMULATION AND ANALYSIS OF THE DEEP-DRAWING PROCESSES

Florica Mioara ŞERBAN

Abstract: *This paper presents an analysis of the deep-drawing process of cylindrical cups with flange using the finite element method. The deep-drawing of sheet metal parts having a good quality is highly influenced by the optimum exploitation of the formability and the control of the technological parameters. In order to avoid the rejection of the finished parts, a detailed investigation of the forming process mechanics is needed. The use of the numerical simulation is justified by its capability to provide useful data related to the deep-drawing processes.*

Key words: *finite element method, deep-drawing, simulation, sheet metal forming, thickness, elastic recovery.*

1. INTRODUCTION

Sheet metal forming is a significant manufacturing process for producing large variety of automotive parts and aerospace parts as well as consumer products. Deformation of sheet materials in the stamping process is classified by the four deformation modes, i.e. deep drawing, stretching, stretch flanging and bending [11].

Deep drawing is one of the widely used sheet metal working processes in the industries, to produce cup shaped components at a very high rate. Cup drawing, besides its importance as forming process, also serves as a basic test for the sheet metal formability. During the course of deep drawing, the following five processes take place [7]: 1) pure radial drawing between the die and blank holder, 2) bending and sliding over the die profile, 3) stretching between the die and the punch, 4) bending and sliding over the punch profile radius, and 5) stretching and sliding over the punch face. Thus, the deep drawing process involves complex deformation mechanisms. The parameters that affect the success or failure of a deep drawing operation are: the punch and die radii, the punch and die clearance, the press speed, the lubrication and the type and the extent of restraint to metal flow material in deep-drawn shapes. Among these, the die shoulder radius [2, 3, 9], punch nose radius [2, 3] and the blank holder force [2, 3, 8, 9, 13] are considered to be the significant parameters in deep-drawing processes. Noticeable differences in forming behavior on the stamping have been observed in the aluminum alloys. The relationship between the material, die design parameters and test parameters versus the deep draw ability has not been well defined [11]. The quality characteristics chosen for the experiment should reflect as accurately as possible the design parameters under study.

Thickness is one of the major quality characteristics in sheet metal formed part [3, 9]. The thickness is unevenly distributed in the part after deep drawing. Generally, the thickness is uniform at the bottom face of the punch, minimum at the punch nose radius and vertical surface, and thicker at the flange area. Existence of thickness variation from the production stage may cause stress concentration in the part, leading to the acceleration of damage. The selection of appropriate process parameters and their combination results in high quality parts.

Finite element (FE) simulation of metal forming process provides an effective means to investigate the relationship between processing deformation and tuning of the production process. Finite element simulation analysis is increasingly used for simulation of deep drawing process [4, 10].

This paper presents an analysis of the deep-drawing process of cylindrical cups with flange using the finite element method.

2. FINITE ELEMENT SIMULATION OF THE DEEP-DRAWING PROCESS OF CYLINDRICAL CUPS WITH FLANGE

The development of finite element method and computer technology gives a realistic image of the process of sheet metal deformation. The costs due to the implementation of a new technology are much reduced by prior determination of the parameters to be used.

There are two primary goals for the use of Finite Element Method (FEM) in analysis of a sheet metal forming process. First analysis aims to reduce the trial and error in tooling and process design, and thereby reduce the material waste and lead times to produce a new part. Second, the analysis aims to influence the design of the desired part for ease of manufacture. Using Dynaform program is possible to achieve the two goals.

2.1 Conditions of numerical simulation

The finite element analysis of deep drawing process for cylindrical parts is carried out with DYNAFORM [18, 19] software. This software package is highly specialized for sheet metal forming in the automotive industry and widely used. DynaForm uses the explicit LSDyna solver. The simulation program allows to easily changing the characteristics of material and other input parameters which makes the program to be used for a wide variety of materials depending on the target.

The material used in the present work for numerical simulations is a deep drawing quality (DDQ) steel. The initial thickness for the blank is 1mm. The deep drawing ratio was defined as [16]:

$$m=d/D \tag{1}$$

where d is the piece diameter d=100mm

D is the blank diameter D=185mm.

A parameter that determines the quality of the deep-drawing parts is the level of the force exerted by the blank holder on the flange. The other important parameter of the process is the punch stroke. Most of the single-action presses used in the automotive industry only allow a constant blank holding force. The level of this force is determined by Eq. (2) [5, 15, 16]:

$$Q = q \cdot A_f \tag{2}$$

where:

Q - blank holding force;

q - holding pressure;

A_f - initial surface of the flange region.

The literature [5, 15, 16] recomends the use of a holding pressure $q = 2 \div 2{,}5$ N/mm^2 in case of sheets made of steel having nominal thickness greater than 0.5 mm.

The initial surface of the flange region was calculated using Eq (3).

$$A_f = \frac{\pi D^2}{4} - \frac{\pi}{4}\left(d_{die} + 2r_{die}\right)^2 \tag{3}$$

where:

d_{die}=d=100 mm; r_{die}=5mm; $A_f \cong 17377\ mm^2$

Making substitutions in Eq.(2), we obtain:

$$Q_{\min} = q_{\min} \cdot A_f = 2 \cdot 17377 = 34754\ N \tag{4}$$

$$Q_{\max} = q_{\max} \cdot A_f = 2.5 \cdot 17377 = 43442.5\ N \tag{5}$$

Numerical simulations were performed for three die radii: r_{die1}=5mm, r_{die2}=8mm, r_{die3}=10mm and three values of the blank holding force namely: Q_1=35000N, Q_2=40000N, Q_3=45000N.

For a realistic simulation of the deep drawing process, besides the geometrical model of the die, the models of the active surfaces of the remaining tools (punch and blank holder) are also needed.

In general, the deep drawing processes can lead to local thickening of the sheet up to 25% of the nominal thickness [6, 12, 14-17]. For this reason it is recommended to offset the die surface at 1.25 * sheet thickness = 1,25*1=1,25mm in order to generate the corresponding surfaces of the punch and blank holder, respectively. The geometric models thus obtained are shown in Figure 1.

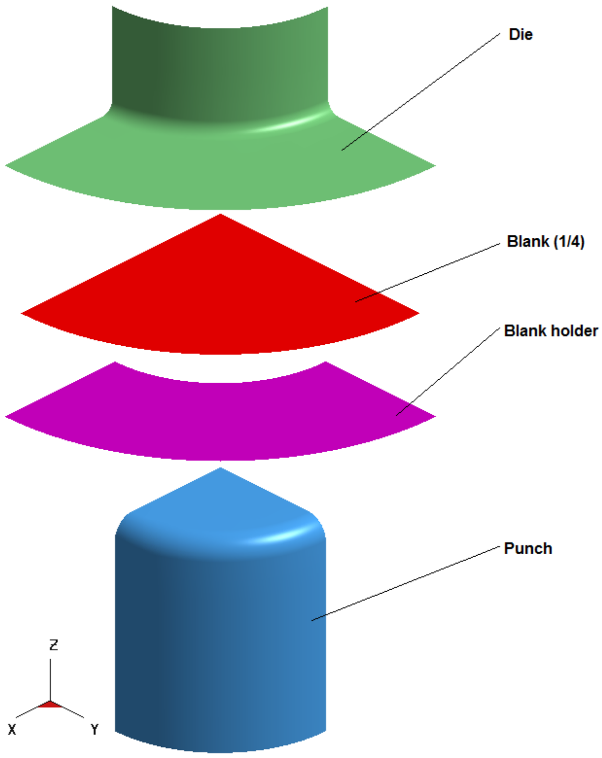

Fig. 1. Tooling used in deep drawing.

The geometry of the blank and the tools is axisymmetric. However, due to the orthotropy of the material behavior [18-20], an axisymmetric 2D analysis cannot be performed. A 3D analysis, modeling only a quarter of the deep drawing process is achieved, which led to a considerable shortening of the computing time. Adequate boundary conditions must be imposed along the symmetry axes. These symmetry axes are defined as the global X and Y axes in the FE mesh; the global Z axis is parallel to the punch displacement direction.

The orthotropy of the sheets has imposed the choice of the material model. This model is based on the following assumptions:

- The reversible component of the strain is described by Hooke's law
- The irreversible component of the strain is described by Barlat 1989 yield criterion and the Hollomon hardening law.

For sheet metal forming processes, the friction influences the material flow and the final strain distribution. The constant friction condition is assumed at all tool interfaces. The friction coefficient at the interfaces between the blank, punch, blank holder, and die is assumed to be 0.1.

The elements used were 4-node Belytschko-Lin-Tsai [18, 20] shell elements, which provide five integration points through the thickness of the sheet metal. The tooling was modeled using rigid surfaces. The active surfaces of the tools were meshed using non-deformable faces.

The positioning of the tools on the blank, taking into account the machine type on which the forming process takes place, is shown in Figure 2.

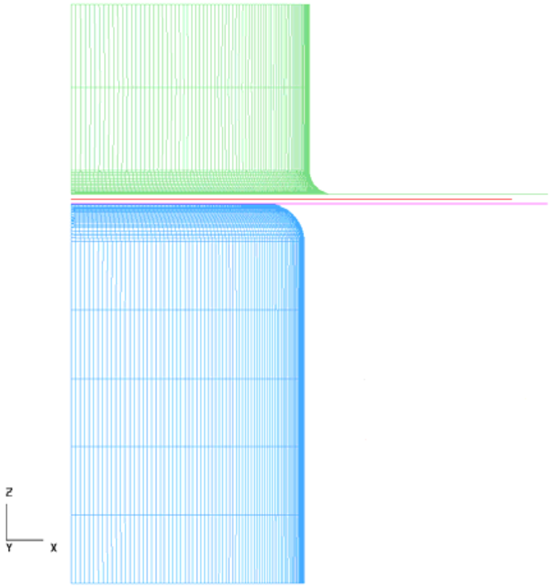

Fig. 2. Finite element model of the deep-drawing of cylindrical cups with flange

2.2 Results of the numerical simulation

In the next will be presented in detail the results obtained for DDQ steel, a blank holder force Q=40000N, the die radius of 5mm and the punch stroke of 45mm. For the other analyzed cases the results are presented in figure 7.

The existence of thickness variation in the deformed part may cause stress concentration and may lead to the damage acceleration. Figure 3 shows the thickness distribution of the simulated sample for a deep drawing ratio of 0.54. The quality of the numerical results are in agreement with the literature [15, 16], which mentions a pronounced thinning tendency of the sheet at the connection with the flat region of the part, and the thickening tendency towards the top of the cup.

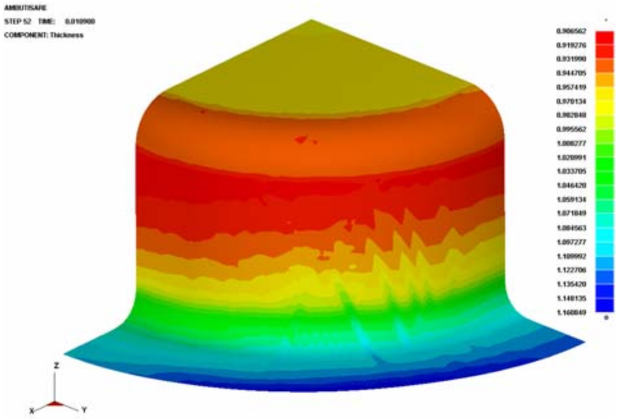

Fig. 3. Simulated deep-drawing process of cylindrical cups with flange (thickness distribution)

To estimate the risk level of forming, the eta/Post postprocessor uses the so-called forming limit curve [1]. This represents the pairs of principal strains (i.e. deformation of the sheet surface) which produce the rejection of the part by necking or cracking. All the points below the curve correspond to strain states technologically admissible, while the points on the curve or above it determine the necking or cracking.

In the left side of the diagram shown in Figure 4 a comparison of the forming state of the drawn part with two such curves is presented. The lower curve represents the forming states with the risk of crack (the possible occurrence of necking) and the curve located above defines the states inducing the crack of the material. It may be noted that the entire part surface is located in the safety area; even if the strains are severe.

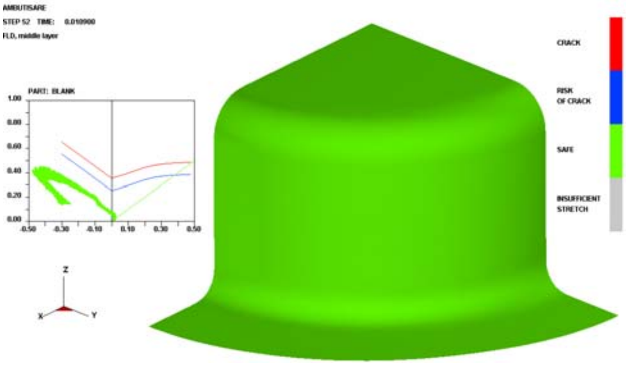

Fig. 4. Evaluation of the risk level of the strains in the final stage of the deep-drawing operation

The variation diagram of the punch load (Fig. 5) shows a maximum of 145 kN. This value is reached at a stroke of about 24 mm. By its aspect, the diagram also agrees with the literature [16].

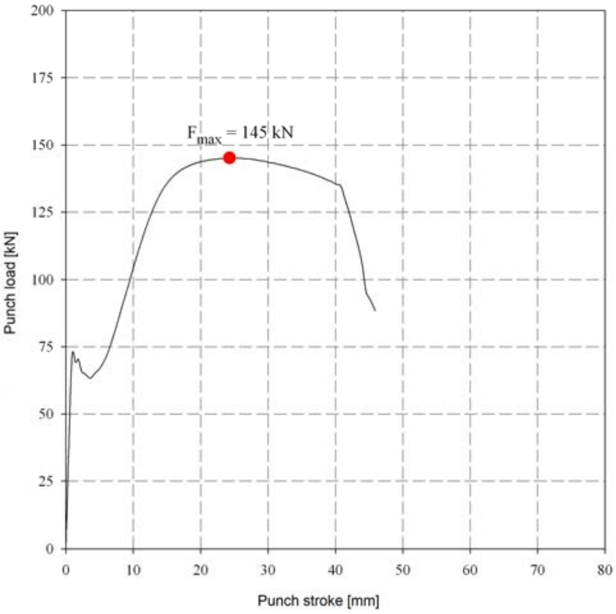

Fig. 5. Punch load versus stroke

2.3. Simulation of the elastic recovery

A very important technological aspect is the elastic recovery after the removal of the part from the deep-drawing die. A pronounced springback of the part determines defects on the finished product affecting its quality.

The simulation of the elastic recovery consisted in suppressing all the tools (die, punch and blank holder), followed by the stress relaxation in the part.

The numerical simulation results of the spring back analysis are shown in figure 6.

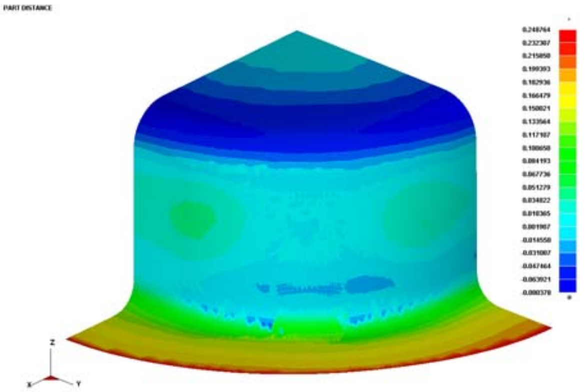

Fig. 6. Dimensional deviations of the part after its relaxation

Using the measuring instruments of the DYNAFORM program the following aspects were found:

- In comparison with the nominal dimensions of the drawn part the springback is reduced. The maximum value is about +0,249 mm corresponding to the flange region (see fig.6) and the minimum value is -0,080 mm. In the cylindrical wall, the deviations do not exceed 0.100 mm.
- Springback mainly affects the outer areas of the part. This is explained by the low level of accumulated plastic strain in these regions during the deep-drawing process.
- In all the areas of the finished part, the elastic recovery is very low. This phenomenon is explained by the very good stretching of these areas during the deep-drawing process.

Figure 7 presents the simulation results for all the analyzed cases. One may be notice an increase of the elastic recovery while increasing the blank holding force. It can be also observed the increase of the elastic recovery with the growth of the die radius.

It can be concluded that the results of the deep-drawing process of cylindrical cups with flange is as satisfactorily as possible.

In the case of the cold formed parts with complex configurations, the relaxation of the internal stresses is not complete. This situation is the result of the non-uniform plastic strain distribution.

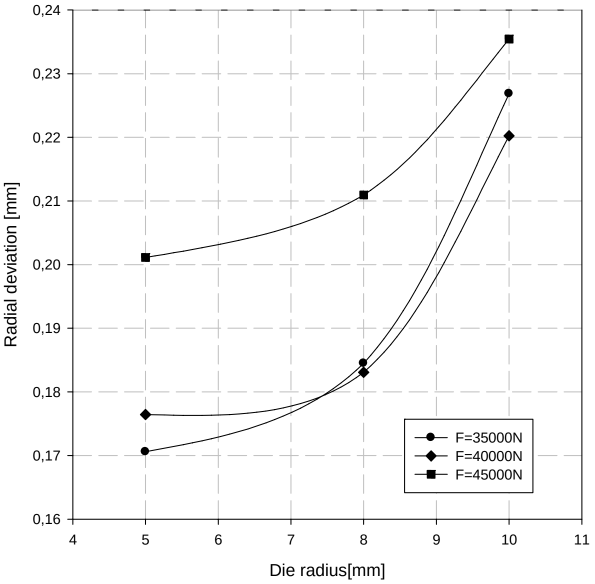

Fig. 7. Elastic recovery

In fact, because of the local incompatibility of the irreversible distortions, a part of the elastic strain remains captive in the material structure. Every further processing that cause changes in the stiffness of the parts can lead to additional spring back.

3. CONCLUSION

The simulation of the deep-drawing process of cylindrical cups with flange showed the power of finite element method in the analysis of this technology. By using the simulation, it was relieved the thickness distribution and the elastic recovery for three blank holder forces and three die radii. The results are in agreement with those presented in the literature.

The simulation results are the base in the future for the design of new equipment with industrial applications.

ACKNOWLEDGMENT

This paper was supported by the project "Development and support of multidisciplinary postdoctoral programmes in major technical areas of national strategy of Research - Development - Innovation" 4D-POSTDOC, contract no. POSDRU/89/1.5/S/52603, project co-funded by the European Social Fund through Sectorial Operational Programme Human Resources Development 2007-2013."

4. REFERENCES

[1] Banabic, D., Dörr, I.R. *Deformabilitatea tablelor metalice subţiri. Metoda curbelor limită de deformare*. Bucureşti: Editura O.I.D.I.C.M., 1992.

[2] Brable, G., Nanu, N., Radu, E. M., *Deep drawing tools and process optimization based on Taguchi and LMecA-Taguchi methods for the compensation of errors generated by springback,* Proceedings of National Conference on Excellence Research−A way to Innovation, Brasov, 2008: 27−29.

[3] Browne, M. T., Hillery, M. T., *Optimising the variables when deep-drawing CRI cups*, Journal of Material Processing Technology, 136: 64−71, 2003.

[4] Desai, S. G., Pardeshi, R.H., Date, P.P., *Study of various initial blank shapes to minimize the earing in the octagonal shaped multistage – formed part using finite element analysis*, Proceedings of the 8th Esaform Conference on Material Forming , Editor: Prof. Dorel Banabic, pp. 325-328, ISBN:973-27-1174-4, Cluj-Napoca, April 2005, The Publishing House Of The Romanian Academy, Bucharest.

[5] Doege, E., Behrens, B.A. *Handbuch Umformtechnik. Grundlagen , Technologien, Maschinen.* Berlin: Springer, 2007.

[6] Iliescu, C. *Tehnologia ştanţării şi matriţării la rece*. Bucureşti: Editura Tehnică, 1977.

[7] Johnson, W., Mellor, P. B., *Engineering plasticity*. Ellis Horwood: Camelot Press, 1983.

[8] Leu, D. K., *The limiting drawing ratio for plastic instability of the cup drawing process*, Journal of Material Processing Technology, 86: 168−176, 1999.

[9] Padmanaban, R., Oliveira, M., Alves, J. L., Menezes, L. F., *Influence of process parameters on the deep drawing of stainless steel*, Finite Elements in Analysis and Design43:1062−1067. , 2007,

[10] Park, D. H., Kang, S. S., Park, S. B., *A study on the improvement of formability for elliptical deep drawing processes*, Journal of Materials Processing Technology, 113: 662-665, 2001.

[11] Raju, S., Ganesan, G., Karthikeyan, R., *Influence of variables in deep drawing of AA 6061 sheet*, Transactions of Nonferrous Metals Society of China, 20:1856-1862, 2010.

[12] Rosinger, Şt. *Procese şi scule de presare la rece. Culegere de date pentru proiectare.* Timişoara: Editura Facla, 1987.

[13] Sheng, Z.Q., Jerathearanat, S., Altan, T., *Adaptive FEM simulation for prediction of variable blank holder force in conical cup drawing*, International Journal of Machine Tools and Manufacturing, 44: 487−494, 2004.

[14] Smith, D.A. (ed.) *Die Design Handbook*. Dearborn: Society of Manufacturing Engineers, 1990.

[15] Suchy, I. *Handbook of Die Design*. New York: McGraw-Hill, 2006.

[16] Tăpălagă, I. ş.a. *Tehnologia presării la rece, vol. I.* Cluj-Napoca: Litografia Institutului Politehnic, 1985.

[17] Tschaetsch, H. *Metal Forming Practise. Processes – Machines – Tools*. Berlin: Springer, 2006.

[18] *** *eta/DYNAFORM. BSE Training Manual (Electronic documentation, – release 5.6.1).* Engineering Technology Associates, 2008.

[19] *** *eta/DYNAFORM. User's Manual (Electronic documentation, – release 5.6.1).* Engineering Technology Associates, 2008.

[20] *** *eta/DYNAFORM. Application Manual (Electronic documentation, – release 5.6.1).* Engineering Technology Associates, 2008.

SIMULAREA NUMERICĂ ŞI ANALIZA PROCESELOR DE AMBUTISARE

Rezumat: Lucrarea prezintă o analiză a procesului de ambutisare de piese cilindrice cu flanŞă folosind metoda elementelor finite. Obţinerea unor produse de calitate corespunzătoare prin deformare plastică la rece (ambutisare) este condiţionată în foarte mare măsură de exploatarea optimă a deformabilităţii semifabricatelor, precum Şi de controlul parametrilor tehnologici. Pentru evitarea apariţiei unor rebuturi, este de dorit cunoaşterea cât mai detaliată a mecanicii proceselor care conduc la atingerea configuraţiei finale a produselor. Recurgerea la simulare numerică se justifică tocmai prin posibilitatea obţinerii unor asemenea informaţii.

Florica Mioara ŞERBAN, PhD. Eng., Technical University of Cluj-Napoca, Department of Manufacturing Engineering, 103-105 Muncii Bvb., 400641 Cluj-Napoca, E-mail: Florica.Groze@tcm.utcluj.ro