ARCHIVES of
Computational Materials Science and Surface Engineering

International Scientific Journal published quarterly by the Association of Computational Materials Science and Surface Engineering

2009 • Volume 1 • Issue 2 • 106-111

Parametric Finite Element Analysis of square cup deep drawing

F. Ayari [a, *], T. Lazghab [b], E. Bayraktar [c]

[a] Laboratory of Mechanics, College of Science and Technology (ESSTT), 1008 Montfleury, Tunis, Tunisia
[b] Research Unity Materials Science, URGM- ENIT Tunis, Tunis, Tunisia
[c] School of Mechanical and Manufacturing Engineering, Supmeca/LISMMA (EA2336)-Paris, France
* Corresponding author: E-mail address: fayza.ayari@gmail.com

Received in a revised form 16.05.2008; published 15.02.2009

ABSTRACT

Purpose: This paper deals with the FEA of the sheet metal forming process that involves various nonlinearities. Our objective is to develop a parametric study that can leads mainly to predict accurately the final geometry of the sheet blank and the distribution of strains and stresses and also to control various forming defects, such as thinning as well as parameters affecting strongly the final form of the sheet after forming process.

Design/methodology/approach: In cold forming (deep drawing) operation, sheet metal is subject to large strains in order to obtain the final desired shape. However, under severe forming conditions the sheet metal may experience some thinning and even some tearing during the process. Parameters of the deep forming process that may contribute to such conditions include, aspect ratio, blank initial thickness, forming temperature, shoulder radii of the die and punch, contact conditions between the blank and the die, holder and punch, punch displacement rate, etc. The work presented in the current paper is a first part study of numerical parametric investigation that is dealing with the most influent parameters in a forming process to simulate the deep drawing of square cup (such as geometric parameters and coefficient of friction). The purpose of the current paper is to conduct a validation study of the FEM model that is used to conduct the study described above. In fact, a 3D parametric FEA model is built using ABAQUS /Explicit standard code.

Findings: A methodology to develop this kind of theoretic resolution is pointed out and has been illustrated for a set of variables. Several 2D and 3D plots, which can be used to predict incipient thinning strengths for sheets with flat initial configuration, have been presented for the various loading conditions.

Research limitations/implications: As it was mentioned above, this paper is the first part of a study of the numerical parametric investigation that is dealing with the most influent parameters in a forming process to simulate the deep drawing of square cup (such as geometric, material parameters and coefficient of frictions).

Keywords: Finite Elements Analysis; Deep drawing; Simulation

Reference to this paper should be given in the following way:

F. Ayari, T. Lazghab, E. Bayraktar, Parametric Finite Element Analysis of square cup deep drawing, Archives of Computational Materials Science and Surface Engineering 1/2 (2009) 106-111.

METHODS OF ANALYSIS AND MODELLING

1. Introduction

The complex deep drawing of thin metallic sheets is widely used during industrial material forming applications. It allows production of thin walled parts with complicated shapes such as automotive panels or structural parts. The process consists of the plastic deformation of an initial at blank subjected to the action of a rigid punch and die while constrained on the periphery by a blank holder. Conventional design processes for sheet metal forming are usually based on a empirical approach. However, due to the requirement of high precision and reliability in shaped parts, these methods are far away from a final and reliable solution. Nowadays, Finite Element Method (FEM) is being gradually adopted by industry to envisage the formability properties of sheet metals. Very useful simulations for deep-drawing processes have been developed by using several commercial codes based on incremental or inverse approaches during the last 20 years [1,2]. Essentially, the incremental codes are based on implicit or explicit methods and take into consideration large elastic-plastic strains and contact conditions with friction between the tools and the sheet. The usefulness of incremental codes has been fully recognized to evaluate the stamping formability by predicting quality defects such as wrinkling, buckling, necking, fracture, and springback permit manufacturers to avoid long and expensive practice procedures. In fact, the metallic material is subjected to large irreversible deformation in sheet forming processes. This leads to high strain localization zones and then internal or superficial micro-defects (ductile damage) [3] This damage causes quality problems such as necking and fracture, leading to process interruptions. For that reason, simulation methods should be used to accomplish precise prediction of the damage mechanism. Therefore affaire, generally coupled approaches are used by including the damage equation with the constitutive equations in that [1-10].

Table 1.
Materials characteristics

	HFS	MS	AL
Yield Stress, MPa	420	173.1	135.3
Young Modules, GPa	211	206	71
Fracture strength, MPa	1350	311.4	277.9
Thickness, mm	1.2	0.78	0.81
Density, kg/m^3	7800	7800	2700

The aim of this study is to evaluate the influence of parameters by means of a FEM analysis of a square box. Deep drawing processes presented in this paper were achieved by using aluminium, mild steel and HFS steel sheets which are commonly used in sheet forming applications. The material and forming characteristics used being listed in Table 1. In this study, a detail analyse was given on the forming of three dimensional shapes with deep drawing process. In fact, the most efficient way to evaluate this type of problem is to analyse the forming step with a FEM code allowing both dynamic and static analysis. In this paper, ABAQUS/Explicit [11] was used and to carry out the static analysis. Since the forming process is essentially a quasi-static problem, the computations with ABAQUS/Explicit can be performed over a sufficiently long time period to render inertial effects negligible. However, this is computationally more expensive and will be prohibitively expensive for simulation of the forming of realistic complex components.

2. Description of the model

In this section, first of all, the structural material of the different parts of the model is expressed and then the stress - strain variables governing the different parts under particular boundary conditions are well described. The material of the blank shown in Figure 1 will form the base of the cup which is in contact with the face of the punch, the die and the blank holder. This material can stretch and slides over the surface of the punch; however, minimal variation in thickness of this material is expected.

During a deep drawing operation, the blank is subjected to radial stresses due to the blank being pulled into the die cavity and there is also a compressive stress normal to the element which is due to the blank-holder pressure. The radial tensile stresses lead to compressive hoop stresses because of the reduction in the circumferential direction.

The flange of the blank attempts to wrinkle because of this hoop stress; however, the blank-holder should prevent occurrence of this from. In fact, the load applied on the blank is modelled as a distributed load on the contact surface blank holder - blank.

The walls of the cup primarily practice a longitudinal tensile stress, as the punch transmits the drawing force through the walls of the cup and through the holder as it is drawn into the die cavity. There is also a tensile hoop stress caused by the cup being held tightly over the punch.

2.1. Concept

The choice of the different geometric dimensions and material properties was conformed to experimental previous data [12]. In fact, the blank is initially square, 150 mm by 150 mm, and thickness is 1.2 mm for the HFS, 0.78 mm for the MS and 0.81 mm for the aluminium material. The rigid die is a flat surface with a square hole 84 mm by 84 mm, rounded at the edges with a radius of 8 mm. The rigid square punch measures 70 mm by 70 mm and is rounded at the edges with the same 10 mm radius. The blank holder can be considered a flat plate, since the blank never comes close to its edges. The geometry of these parts is illustrated in Figures 1 and 2. The rigid surfaces are offset from the blank by the half of the thickness of the blank to account for the shell thickness as ABAQUS/Explicit automatically takes the shell thickness into account during the contact calculation.

A mass of 0.65 kg is attached to the blank holder, and a concentrated load of 19.6 kN is applied to the contact surface blank - holder. The blank holder is then allowed to move only in the vertical direction to accommodate changes in the blank thickness. The friction coefficient between the sheet and the punch is taken to be variable from 0.01 to 0.125, and also between the Blank and the Punch varies from 0.01 to 0.25. In fact, formerly, it was confirmed that the friction coefficient between contact surfaces has an important effect in the forming process [12,13].

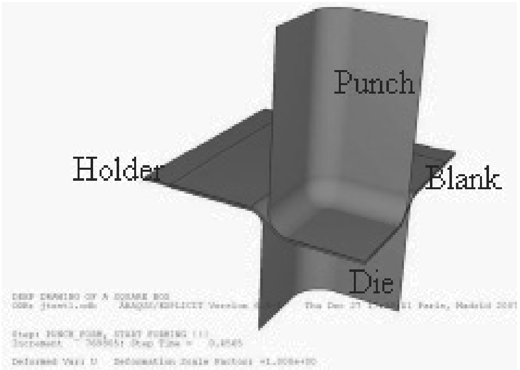

Fig. 1. Different parts in the assembly model

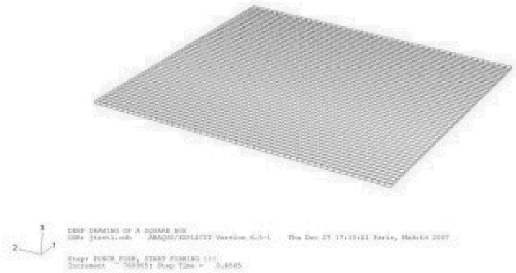

Fig. 2. Blank part at initial configuration

The simulated punch velocity is kept constant and equal to 1.66 mm/sec while the considered minimum nodal distance is less than the blank thickness. The blank is made either of aluminium, HFS and Mild steel.

The relation between true stress and logarithmic strain, were given with the constitutive equations for these materials in the reference of Bayraktar and Altintas (1996) [12]. The stress-strain behaviour is defined by piecewise linear segments matching the Ramberg-Osgood curve up to a total (logarithmic) strain level of 107%, with Mises yield, isotropic hardening, and no rate dependence except for the HFS material.

Given the symmetry of the problem, it is sufficient to model only a one-quarter sector of the box. However, we have employed a one-quarter model to make it easier to visualize. Only 4-nodes are used with three-dimensional rigid surface elements (type R3D4) to model the die, the punch, and the blank holder. The blank is modelled with 8-nodes, linear finite-strain shell elements (type SC8R).

The computer time involved in running the simulation using explicit time integration with a given mesh is directly proportional to the time period of the event, since the stable time increment size is a function of the mesh size (length) and the material stiffness. Thus, it is usually desirable to run the simulation at an artificially high speed compared to the physical process.

If the speed in the simulation is increased too much, the solution does not correspond to the low-speed physical problem; i.e., inertial effects begin to dominate. In a typical forming process, the punch may move at speeds on the order of 1 m/sec, which is extremely slow compared to typical wave speeds in the materials to be formed. (The wave speed in steel is approximately 5000m/sec.) In general, inertia forces will not play a dominant role for forming rates that are considerably higher than the nominal 1 m/sec rates found in the physical problem.

In the results presented here, the drawing process is simulated by moving the reference node for the punch downward through a total distance of (11- 15- 30 and 40 mm 6.626506, 9.036145, 18.072289 and 24.096386. In this analysis we used the technique of mass scaling to adjust the effective punch velocity without altering the material properties.

3. The FEM model

The numerical FEM simulations of the square cup deep drawing process were conducted with the ABAQUS/Explicit commercial code. The configurations of the punch, the die, the blank and the holder are illustrated in Figure 1. The movement of the punch was defined using a pilot node. This node was also employed to obtain the drawing force during the simulation. The distributions of the von Mises stresses at two different stages are illustrated in Figure 3. Thickness variation of the blank section is illustrated with Figure 4 and the principle equivalent plastic strain is also done by the Figure 5. For the sake of the simplicity, the remaining of the FE findings are presented and discussed in the next section to compare the various results in detail.

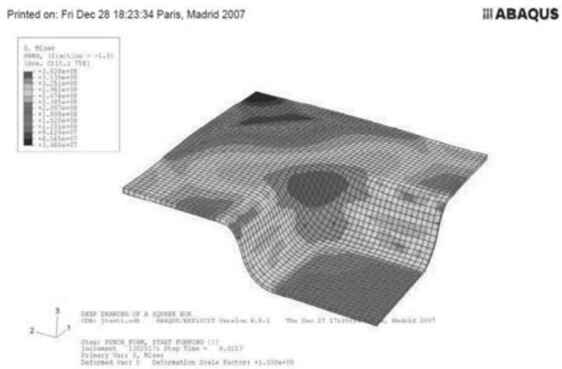

Fig. 3. Von Mises stress distribution for the MS Blank at the end of the step with a travel of 15 mm

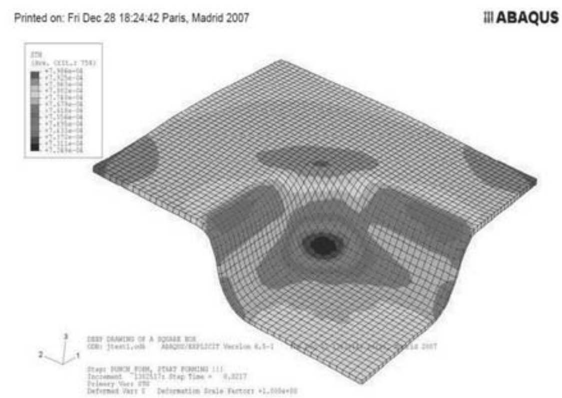

Fig. 4. Thickness section distribution for the MS Blank at the end of the step with a travel of 15 mm

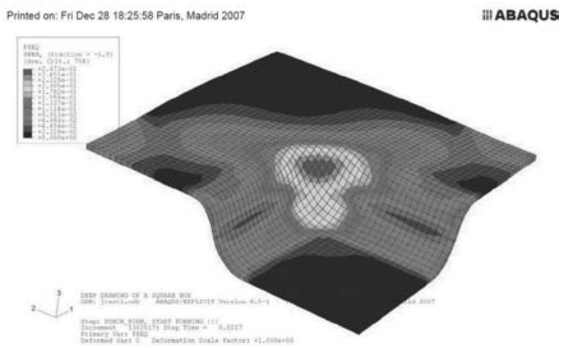

Fig. 5. Principle equivalent strain distribution for the MS Blank at the end of the step with a travel of 15 mm

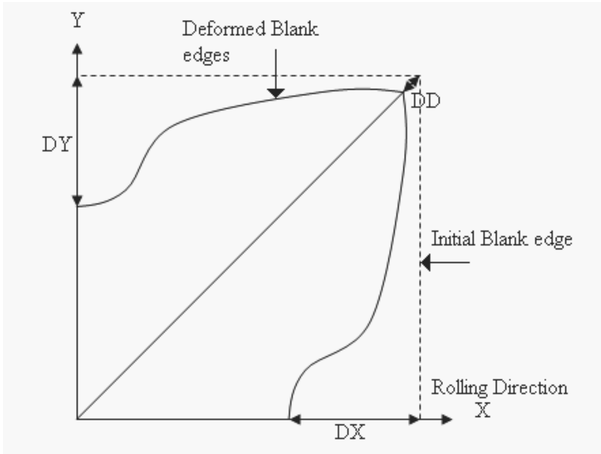

Fig. 6. Schematic description of the validation geometric parameters [12]

4. Comparison between experimental and numerical results and validation

To check the validity of the results computed by deep drawing simulations, the influence of some important numerical parameters was investigated. These parameters refer to the steel sheet constitutive law, the FE mesh, the FE analysis and the friction behaviour.

4.1. General displacements

Hadfield steel, Mild steel and Aluminium materials are used in this study for comparison. The experimental results listed in Table 2 for the square cup drawn to a punch travel of 15mm and 30 mm for Hadfield steel and also 15 mm and 40mm for Mild steel sheets specimens, than 11mm and 30 mm for the aluminium sheets specimens. The principle strains (ε_1,ε_2) which were measured in the rolling direction DX, diagonal direction DD and transverse direction DY respectively Figure 6. Comparative numerical and experimental results for 3D direction corner nods were given Table 2 and Figure 6 [8].

Table 2.
Results of experimental and numerical study [12]

Travel	Dir	HFS		MS		Al	
		Exp	Num	Exp	Num	Exp	Num
15	DX	5.9	5.89	7.0	7.07		
	DY	2.7	2.77	3.9	3.7		
	DD	6.1	5.9	7.1	7.06		
30	DX	10.0	10.1				
	DY	7.0	6.8				
	DD	10.0	10.1				
40	DX			28.1	28.7		
	DY			15.1	14.89		
	DD			28.5	28.6		
11	DX					3.8	3.709
	DY					2.3	1.993
	DD					3.9	3.713

The numerical results presented in the Table 2 are obtained with the following coefficient of friction 0.02 for the Blank Holder contact, 0.02 for the Blank - Die contact, 0.25 for the Punch Blank contact and than 0.03 for the global contact surfaces. In fact, these coefficients are chosen to simulate the real contact surfaces of the experimental case; as before each experiment both sides of the Blank sheet surface were wiped with a paper towel dipped in the lubricant and they were kept in a vertical position for 30 minutes [12,13]. The second validation to be considered in this study consists of the comparison between the principle strains (ε_1, ε_2, ε_3) for a punch travel of a 15 mm, 30 mm and 40 mm for the HFS.

In the Figure 7 the distance in the rolling direction versus the principal strains using numerical and experimental results of the square cup deep drawing are presented for the Mild steel sheet Blank with a travel punch distance of 15mm. In the same way principal strains collected for the transverse direction and the diagonal direction by Figures 8 and 9.

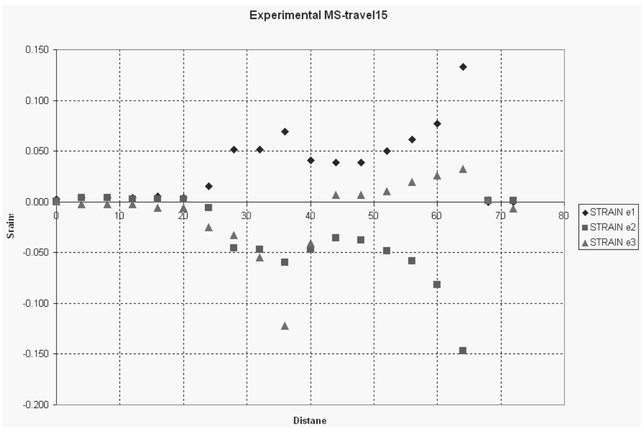

Fig. 7. Principal strains in the rolling direction for MS steel with a travel punch distance of 15 mm experimental results

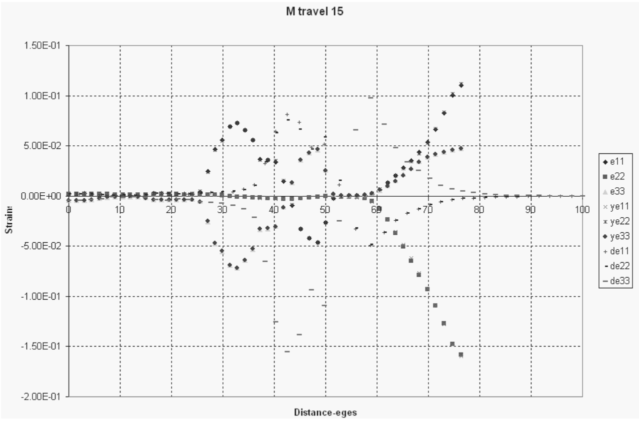

Fig. 8. Principal strains for MS steel with a travel punch distance of 15 mm- numerical results

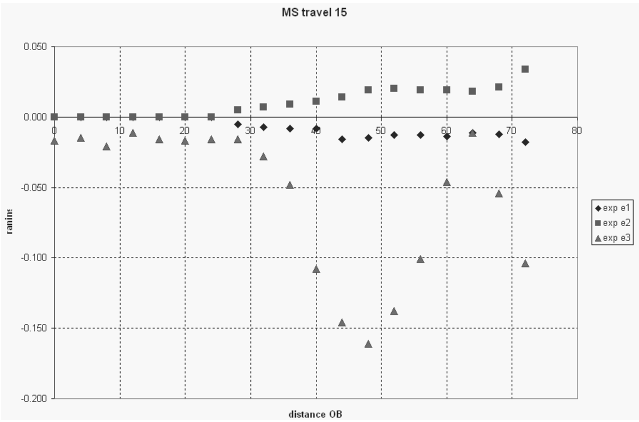

Fig. 9. Principal strains in the transverse rolling direction for MS steel with a travel punch distance of 15 mm

It is important to show that the numerical and experimental strain variation profile is almost similar for the different directions.

Amplitude of the major error between numerical and experimental values is about 33 % for the MS, this fact is expected. Thus, the different properties of the materials presented in Table 1 are the result of the mean values calculated from different direction of the materials and as it is known that the 3D drawing process are sensitive to the anisotropy.

In Table 2 the amplitude of the earring profile is larger for the HFS steel then the MS material for the same travel distance. The earring profile being closely linked to the coefficient of friction between different contact surfaces. Thus, study of the friction coefficient will be treated in the following section.

4.2. Friction coefficient

Friction is one of the most important parameters, which affects the material flow and the required load in both the bulk forming [13,14,15,16] and sheet forming processes [14]. Friction has both positive and negative roles in metal forming. There are numerous instances where friction opposes the flow of metal in forming processes.

However, there are also several instances where the forming process is made possible by friction such as rolling, drawing of tubes with a moving mandrel and Masllenikov's technique and also the same for drawing operations of the very deep cups and striking examples of friction-assisted metal forming processes. For example rolling is a case of a forming process that would not exist if not for friction [15].

In order to study the influence of the friction coefficient on the displacement corner's blank, several FE analyses were performed for different values of "c". The results obtained from the simulations were illustrated in Table 3. In fact, the value of "c" has not much influence at the early stages of the process. However, at the greater punch travels, the friction affects strongly the required displacement edges.

Table 3.
Results of numerical study influence of coefficient of friction for the edges displacements

C-BH	C-BD	C-BP	C-Gl	DX	DD	DY
0	0	0	0	3.72	1.98	3.7
0.04	0	0.0162	0	3.66	1.9	3.66
0.04	0.04	0.16	0	3.70925	1.9935	3.7138
0.02	0.02	0.25	0.03	3.6842	1.987	3.6854
0.02	0.02	0.25	0.03	3.68	1.99	3.68085
0.02	0.02	0.032	0.03	3.696	1.989	3.69549

C-BH: Blank Holder coefficient of friction
C-BD: Blank Die coefficient of friction
C-BP: Blank Punch coefficient of friction
C-Gl: coefficient of friction for the general contact

5. Conclusions

The main features of the results were reported in this paper regarding the effect of material and forming characteristics on the simulation of the deep-drawing of square cups for the different materials by using the explicit FE code ABAQUS and were taken also into account of the similar remarks reported in the former paper [8] pertaining to the simulation of the deep-drawing of square cups of the same materials.

The verification procedure reported in detail in [13] and outlined here allows for optimal sheet metal forming simulations for simulating materials and forming parameters ranging within the limits indicated in Table 2 and Figures 3 and 4 from reference [13]. For the materials and geometries examined here, the strain distributions, the punch travel curves and the macroscopic FE mesh deformation modes were very well verified through the FE modelling of the deep-drawing process, with the simulating punch velocity, equal to 1.66 mm/sec and the simulating blank material density.

Displacements and strains distributions are the real experimental values of blank deformation. Coulomb friction coefficient at the punch blank interfaces is of 0.25.

The blank displacement predicted by the numerical models was generally in fair agreement with experimental results (Table 1); the influence of numerical parameters was relatively small. In fact, for the coefficient of friction parameter, the influence of numerical parameters on the blank displacement of the edges was summarized in Table 3. Thus, the prediction of the earring profile was in agreement with experimental results for both MS and HFS materials.

The weak anisotropy for aluminium material was assumed to be at the origin of the poor accuracy of the numerical results and explains the small difference in the shape of the curves for different blank-edges displacements. The earring profile prediction is clearly very sensitive to mechanical behaviour of the materials. It appears also that the influence of the coefficient of friction for contact surfaces is relatively small in comparison with other parameters like the constitutive laws of the simulated materials.

Furthermore, the shape of the earring profile used to be almost influenced by the constitutive low and other numerical parameters that will be the aim of extension work of this first part as a parametric study.

This conclusion is only valid for the constitutive lows considered as anisotropic. For higher anisotropic materials, the influence of the constitutive model would be predominant.

References

[1] J.L. Batoz, H. Naceur, Y.Q. Guo, Formability Predictions in Stamping and Process Parameter Optimization Based on the Inverse Approach Code Fast-Stamp, Proceedings of the 6th International Conference “Numerical Simulation of 3-D Sheet Metal Forming Processes - Verification of Simulation with Experiments” Numisheet 2005, Detroit, 2005, 831-836.

[2] J.P. Fan, C.Y. Tang, C.P. Tsui, L.C Chan, T.C. Lee, 3D finite element simulation of deep drawing with damage development, International Journal of Machine Tools and Manufacture 46 (2006) 1035-1044.

[3] Y.Q. Guo, Y.M. Li, F. Bogard, K. Debrey, An efficient pseudo-inverse approach for damage modelling in the sheet forming process, Journal of Materials Processing Technology 151 (2004) 88-97.

[4] Z.Y. Cai, M.Z. Li, Multipoint forming of three-dimensional sheet metal and the control of the forming process, International Journal of Pressure Vessels and Piping 79 (2002) 289-296.

[5] J.L. Chaboche, G. Cailletaud, Integration methods for complex plastic constitutive equations, Computer Methods in Applied Mechanics and Engineering 133 (1996) 125-155.

[6] M. Colgan, J. Monaghan, Deep drawing process: analysis and experiment, Journal of Materials Processing Technology 132 (2003) 35-41.

[7] L. Duchêne, A.M. Habraken, Analysis of the sensitivity of FEM predictions to numerical parameters in deep drawing simulations, European Journal of Mechanics A 24 (2005) 614-629.

[8] F. Fereshteh-Saniee, M.H. Montazeran, A comparative estimation of the forming load in the deep drawing process, Journal of Materials Processing Technology 140 (2003) 555-561.

[9] R. Padmanabhana, M.C. Oliveiraa, J.L. Alvesb, L.F. Menezesa, Influence of process parameters on the deep drawing of stainless steel, Finite Elements in Analysis and Design 43 (2007) 1062-1067.

[10] K. Saanouni, K. Nesnas, Y. Hammi, Damage modelling in metal forming processes, International Journal of Damage Mechanics 9/3 (2000) 196-240.

[11] ABAQUS version 6.4, ABAQUS, Inc., Hibbit, Karlsson & Sorenson.

[12] E. Bayraktar, S. Altintas, Square cu deep drawing and 2D draw bending analysis of Hadfield steel, Journal of Materials Processing Technology 60 (1996) 183-190.

[13] F. Fereshteh-Saniee, H.Bayateefar, Experimental comparison of different friction tests in bulk metal forming, Proceedings of the 7th International Conference “Technology of Plasticity”, Yokohama, 2002, vol. 1, 91-96.

[14] F. Fereshteh-Saniee, I. Pillinger, P. Hartley, Friction modelling for the physical simulation of the bulk metal forming processes, Proceedings of the International Conference “Advances in Materials and Processing Technology” AMPT2003, Dublin, 2003, vol. 1, 543-546.

[15] R.H. Wagoner, J.L. Chenot, Fundamentals of Metal Forming, Wiley, New York, 1997.

[16] Y. Yamaguchi, N. Takakura, M. Fakuda, Attempts to facilitate low volume production of soft aluminium cups with large draw ratios by deep drawing based on Maslennikov’s technique, Journal of Mechanical Working Technology 2 (1979) 357-366.