TECHNICAL TRANSACTIONS 11/2017
CZASOPISMO TECHNICZNE 11/2017

MECHANICS

DOI: 10.4467/2353737XCT.17.202.7431

Patryk Różyło (p.rozylo@pollub.pl)
Department of Computer Modelling and Metal Forming Technologies, Faculty of Mechanical Engineering, Lublin University of Technology

Łukasz Wójcik
Department of Machine Design and Mechatronics, Faculty of Mechanical Engineering, Lublin University of Technology

Comparative analysis of the experimental and numerical research of the stamping process of axially-symmetrical elements

Analiza porównawcza badań doświadczalnych i numerycznych tłoczenia elementów osiowo-symetrycznych

Abstract

The paper presents experimental and numerical results of the stamping process of an axially-symmetrical element. The subject of the study was a sample made of a 0.47 mm thick aluminium alloy Al-1100 sheet. Experimental studies were conducted on a universal testing machine, while numerical simulations were carried out on two different numerical software programs Abaqus® and Deform-3D. The results obtained from the numerical analysis allowed to develop a numerical model of the stamping process.

Keywords: stamping process, numerical and experimental analysis, Abaqus, Deform-3D

Streszczenie

W artykule przedstawiono wyniki badań eksperymentalnych i numerycznych procesu tłoczenia elementu osiowo-symetrycznego. Przedmiotem badań była próbka wykonana ze stopu aluminium Al-1100 o grubości 0,47 mm. Badania doświadczalne przeprowadzono na uniwersalnej maszynie wytrzymałościowej, natomiast symulacje numeryczne zrealizowano w dwóch różnych programach komputerowych Abaqus® oraz Deform-3D. Wyniki uzyskane z analizy numerycznej pozwoliły na opracowanie numerycznego modelu procesu tłoczenia.

Słowa kluczowe: proces tłoczenia, analizy numeryczne i doświadczalne, Abaqus, Deform-3D

1. Introduction

It is common to reduce costs and shorten the design time of metal sheet processing. The designing of sheet metal stamping is a complicated and time-consuming process. The limitations that negatively influence the optimisation of the element shaping processes with a non-expandable surface are complex experimental research, which generates high costs. The costs relate to the implementation of often complex tools. Therefore, methods are sought that would eliminate experimental research. The most commonly used are computer (numerical) techniques, assisting the design of metal forming processes (finite element method or finite volume method). The application of numerical calculations and simulation has allowed for costs and time reduction of the design and preliminary testing of stamping processes. Simulation software is mainly used to predict the direction of the flow of metal, of stress distribution analysis, deformation, temperature or possible defects.

The sheet stamping process is the most common process used to shape thin-walled components. A detailed classification of the stamping process is presented in [1]. The stamping process involves converting the flat sheet portion into a non-expandable surface element. In the stamping process, axially-symmetric [2], rectangular [3] and complex shaped drawpieces are produced.

During the stamping of thin metal sheets, radial stresses – and the peripheral tensile – compression occurs in the formed flanges. It is assumed that the process is carried out in a two-dimensional stress state.

The analysis of the stamping process is based on determining the occurring phenomena and interaction between the drawing and stretching areas. The analysis of the phenomena occurring during sheet metal stamping is described in publication [4-7].

The stamping process is burdened with two major constraints that negatively affect the shaping of the sheet. The most commonly occurring phenomena, restricting the stamping process, are the corrugating of the drawpiece flange and bursting of the cylindrical part of the drawpiece wall. The limitations causing drawpiece or tool damage are described by the researchers in publication [8].

2. Subject of the study

The subject of the study was a disc-shaped aluminium alloy sample. The test sample was characterised by a fixed diameter of 50 mm and a thickness of 0.47 mm. The graphical representation of the test sample is shown in Figure 1.

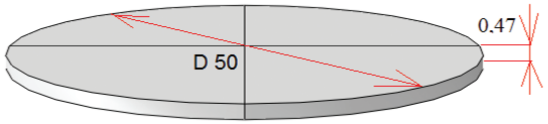

Fig. 1. Geometrical parameters of the specimen

The test samples were characterised by specific material properties, consistent with the definition of the elastic-plastic model. All experimentally determined material characteristics required the correct implementation of the dynamic issue in the FEM analysis, which are presented in Table 1.

Table 1. Material properties of the test sample (own research)

Al-1100 aluminium alloy material properties					
Density $[kg/m^3]$	**Young Modulus** [MPa]	**Poisson's ratio** [-]	**Yield point** [MPa]	**Tensile strength** [MPa]	**Elongation** [%]
2700	70000	0.33	85	115	2

3. Experimental studies

Experimental studies were carried out on the universal testing machine Instron 3369. In order to properly carry out the stamping process, the test specimen was suitably positioned in the lower die. A universal testing machine is equipped with a guideway at the upper part of the die, which provides axial guidance of the stamp. No initial pressure was applied to the test sample. As part of the research, load-displacement characteristics of the stamp have been determined. The process was carried out with a constant traverse speed of 1.67 mm/s. The duration of the analysis was 38.92 s, which was related to the displacement of the stamp by 65 mm. The test stand is shown in Figure 2.

Fig. 2. Testing equipment set-up for the stamping process

The experimental studies allowed obtaining the drawpiece, which was compared with the numerical model. In regard to the conducted test, further validation with computer simulations based on finite element method in two independent environments was needed.

4. Numerical analysis

The numerical analyses were conducted by the use of two independent software – Abaqus and Deform-3D. The key stage in the preparation of the process was the appropriate definition of the material model based on experimental material parameters. The material model had elastic-plastic properties. Geometrical parameters were mapped from the measurements obtained from actual test assemblies. The general contact between cooperating tangential and normal surfaces was determined. A permanent friction model corresponding to the contact conditions of aluminium alloy and steel was adopted. The boundary conditions relevant to the experimental study are shown in Figure 3.

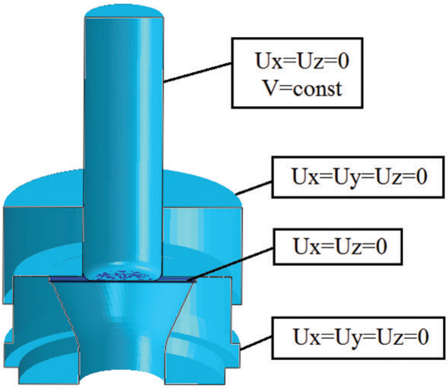

Fig. 3. Boundary conditions of the numerical model

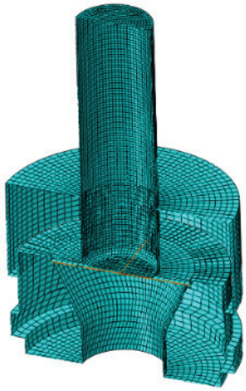
Fig. 4. Discrete model

The discretisation process was based on the use of two types of elements – the C3D8R (deformable) for the test sample in which simulation results were possible to obtain, and the R3D4 coating (non-deformable) for the remaining parts [9]. The number of finite elements of C3D8R type was equal to 25 000 (mesh density – 0.5 mm), while for R3D4 – 30000 (mesh density 1 mm). The adaptive mesh reconstruction algorithm was used. Adequate numerical methods were used for both software environments. The graphical representation of the discrete model obtained in the Abaqus program is shown in Figure 4.

Numerical analysis was based on the solution of the non-linear dynamic issue, under exactly the same conditions as the experimental study.

5. Testing results

The purpose of the numerical and experimental studies was to obtain the stamping process characteristics of the research system. The graphical presentation of the initial stages of the FEM simulation process is shown in Figure 5.

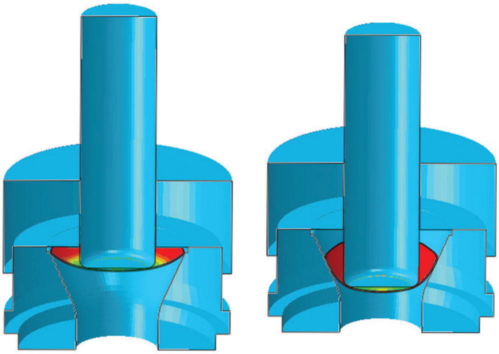
Fig. 5. Graphical presentation of the initial stages of the stamping process

As part of the obtained analysis, the actual form of the drawpiece has been obtained from the flat aluminium alloy sample, as well as from the performed simulations. The approximate drawpiece's shapes, presented in Figure 6, demonstrate the high quality of numerical models.

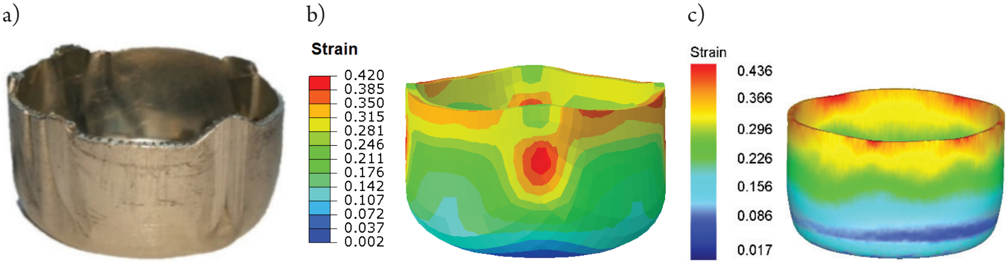

Fig. 6. Graphical representation of drawpieces: a) actual object; b) Abaqus numerical model; c) Deform-3D numerical model

The actual drawpiece is similar to the one obtained by the Abaqus numerical model. The drawpiece obtained from Deform-3D is slightly different from the other cases. Regarding the obtained results of strain level, a high convergence between Abaqus and Deform-3D software was demonstrated.

The conducted studies enabled to derive the necessary characteristics of the system, based on which the convergence of the research results was estimated. Figure 7 shows the stamping process characteristics.

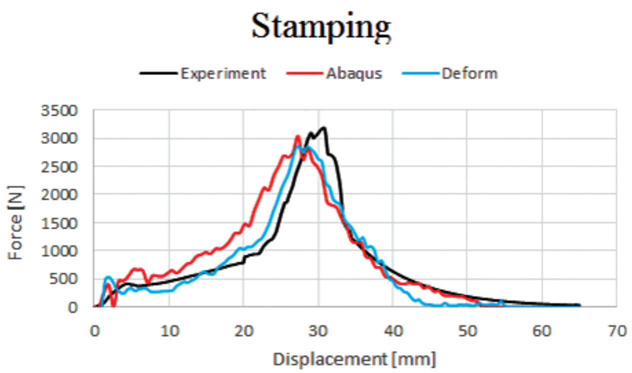

Fig. 7. Comparison of the characteristics of the stamping process

The obtained results, presented in the graph above, constitute about the high quality of the performed tests and prepared numerical models. Both experimental and simulation analysis have shown that the greatest force in the stamping process is approximately 30 mm in the displacement of the stamp. A detailed breakdown of the maximum force values is shown in Table 2.

The results shown in Table 2 exhibit a high convergence of the test outcome. The difference in force obtained by numerical tests in Abaqus relative to the actual sample is only 4.6%. The corresponding difference in Deform-3D is exactly 10.4%.

Table 2. Comparison of the results of maximum stamping forces

Experiment [N]	**Abaqus** [N]	**Deform** [N]	**Difference Abaqus – EXP** [%]	**Difference Deform – EXP** [%]
3185.4	3038.7	2853.7	4.6	10.4

6. Conclusions

The analysis of actual and numerical studies of the stamping process allowed determining the consistency of the obtained results. The results showed a high convergence between computer simulations and experimental studies. It has been demonstrated that there are advanced numerical computation programs that allow for a greater consistency of results – Abaqus and lower – Deform-3D within a given type of issue. The use of numerical calculations is supported by the use of several independent validation capabilities in regard to the actual experimental research. The Abaqus has generated better quality results than Deform-3D. The discrepancy between the Abaqus and the experiment is 4.6%, and between the Deform-3D and the experimental is 10.4%. The quality of the obtained results constitutes about properly prepared numerical model. The overall assessment of the quality of the research results constitutes a well-prepared numerical model.

References

[1] Romanowski W.P., *Poradnik obróbki plastycznej na zimno*, WNT, Warszawa 1976.

[2] Mori K., Nishijima S., Tan C.J., *Two stage cold stamping of magnesium alloy cups having small corner radius*, International Journal of Machine Tools and Manufacture, 49, 2009, 767-772.

[3] Chen F., Huong T., Chang C., *Deep drawing of square cups with magnesium alloy AZ31 sheets*, International Journal of Machine Tools and Manufacture, 43, 2003, 1553-1559.

[4] Pereira M., Rolfe B., *Temperature conditions during cold sheet metal stamping*, Journal of Materials Processing Technology, 214, 2014, 1749-1758.

[5] Vrolijk M., Lorenz D., Porzner H., Holecek M., *Supporting lightweight design: virtual modeling of hot stamping with tailored properties and warm and hot formed aluminum*, Procedia Engineering, 183, 2017, 336-342.

[6] Jin J., Wang X., *A single-step hot stamping-forging process for aluminum alloy shell parts with nonuniform thickness*, Journal of Materials Processing Technology, vol. 228, 2016, 170-178.

[7] Zhou J., Wang B., Lin J., Fu L., *Optimization of an aluminum alloy anti-collision side beam hot stamping process using a multi-objective genetic algorithm*, Archives of Civil and Mechanical Engineering, Vol. 13, 2013, 401-411.

[8] Isik K., Silva M., Tekkaya A., Martins P., *Formability limits by fracture in sheet metal forming*, Journal of Materials Processing Technology, 214, 2014, 1557-1565.

[9] Zienkiewicz O.C., Taylor R.L., *Finite Element Method*, Vol. 2: *Solid Mechanics*, 5th ed., Elsevier, 2000.