Journal for Technology of Plasticity, Vol. 37 (2012), Number 1

FEM SIMULATIONS OF AN AXI-SYMMETRICAL ELEMENT STRAIN STATE

*Šibalić Nikola, Vukčević Milan, Savićević Sreten, Janjić Mileta**

University of Montenegro, Faculty of Mechanical Engineering Podgorica, Montenegro

ABSTRACT

Development of computer technique and software has enabled a quick and easy simulation as well as a relatively simple determination of the needed parameters of a plastically deformed body. In that way, production is stimulated and improved, a rapid development of new products is enabled. In order to give relevant data, it is necessary to connect three methods: numerical, theoretical and experimental. As it is very important to know strain state parameters in deformation processes, a numerical experiment for axial-symmetrical workpiece at bulk metal forming is carried out to determine these parameters. Simulation is carried out by applying DEFORM-2D commercial software package based on the finite elements method. A graded die shape is adopted being convex in its upper part and concave in its lower part, die and workpiece model dimensions are optimized. Simulating the process, it is possible to perform by strain discretization and determine strain parameters. To be able to compare the results obtained, node points arranged along the axial section of workpiece model are adopted to be followed in the course of experiment and to determine their deformation values. Numerical experiment consists of three stages. In the first stage, displacement points are determined at the end of the process for the observed node points, and strains are determined by known relations. In the second stage, displacement of node points determined per discretization phases and strains based on them are determined. Strains at the end of process are obtained by summing strains per phases. In the third stage in the observed points, strains are obtained from the DEFORM-2D software package. The required parameters are graphically interpreted in the paper. Analysis and comparison of the results obtained are made as well.

Keywords: *strain, numerical experiment, simulation, FEM, DEFORM-2D.*

1. INTRODUCTION

The development of modern computer systems has offered to users wide possibilities of projecting and simulating bulk metal forming process, analysis and comparison of the obtained results, as well as an insight into the state of all activities providing immediate and completely reliable data

---

**Corresponding author's email: mileta@ac.me*

on all parameters included in deformation process. The best way is testing on real object, being quite expensive one and demanding a lot of time for research, so numerical experiments that are cheaper and more rapidly developed have been more applied lately. The most widely applied tool for numerical simulation of bulk metal forming processes is the Finite Elements Method (FEM). Due to a rapid development of computer technique, a number of commercial software packages have been developed lately, based on finite element method, for solving problems in metal forming processes. DEFORM-2D whose producer is *Scientific Forming Technologies Corporation* (SFTC), in which numerical simulation is carried out and the results obtained are presented in this paper, will be used [1,2,3,4,5].

2. INPUT PARAMETERS FOR NUMERICAL SIMULATION

Data related to aluminium alloy *AlMgSi0.5*, pressed at temperature of hot forming $T=440\ ^oC$ are used in numerical experiment. Step-shape tool (Figure 1.), consisting of two dies, upper and lower ones, is adopted. The upper die is convex consisting of two height levels, whereas lower one is concave, also consisting of two height levels. The fallowing data are adopted in the paper:

- Deformation is obtained by a small constant forming velocity of $v=2$ *mm/s*.
- Hardening curve expressed as: $\sigma_e = C \cdot \varphi^n$ *kN/cm*^{2}, where are: $c=30.34434$ *- Constant,* $n=0.097808$ *- Strain exponent* for *AlMgSi0.5* aluminum alloy, and temperature $T=440\ ^oC$.
- Friction factor is $m=0.114$.

The initial form of the workpiece is cylindrical, of diameter $d_0=33.56$ *mm*. Height h_0 is determined by the conditions of constant working-pieces volume before and after process for adopted die dimensions given in Figure 1. and it amounts to $h_0=32.87$ *mm*. An adopted mesh of a half of axial-symmetrical initial form of the workpiece model is given in Figure 2. Total number of node points is 154. Point coordinates whose displacement will be followed in numerical experiment and whose strain state parameters will be determined are on nodes of the adopted mesh (Figure 2.).

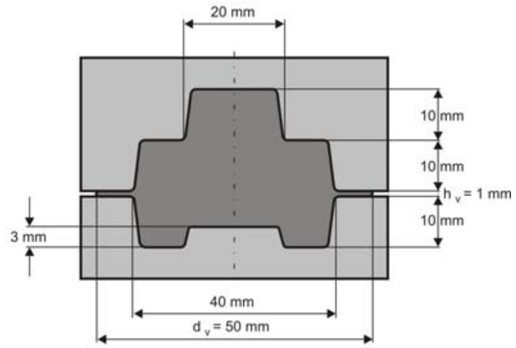

Fig.1 - Workpiece model within die

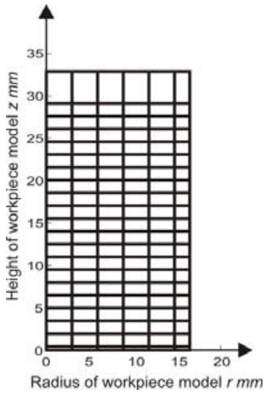

Fig 2 - Adopted mesh of a half of axial-symmetrical workpiece model

3. NUMERICAL SIMULATIONS

3.1. Numerical experiment for continual process

To carry out numerical simulation, it is necessary to input some known data into DEFORM *Pre Processor* module. In *Simulation Control* sub-module, data related to the choice of system units, type, geometry illustration, step number, step recording, stroke value per step, and process stoppage values are input (Table 1.). This study's aim is to get metal flow and distributions of equivalent stress on some special sections such as longitudinal and transverse sections under processing tube tension-reducing. Data related to workpiece model material, i.e. values of hardening curve *Constant* (C) and *Strain exponent* (*n*) are input into sub-module *Material*.

Table 1: Simulation Control

Main	Units	☑SI	
	Geometry	☑Axisymmetric	
	Type	Incremental	
	Mode	Deformation	
Step	Number of Simulation Steps	1000	
	Step Increment to Save	10	
	Primary Die	1 - Upper die	
	With Equal Die Displacement	0.1 mm	
Stop	Primary Die Displacement	0	14.87 mm

By option *Insert* o*bject*, a new object is added and data related to geometry of upper and lower dies, workpiece model, and the upper die movement speed values are input (Table 2.). Data related to die and workpiece model, as well as friction factor values are input into sub-module *Inter-Object* (Table 3.). After inputting data and forming mesh of finite elements, data base is generated in sub-module *Database Generation* for initial step marked as -1. Generating database, all the necessary conditions to carry out deformation process analysis are acquired, namely by entering *Simulator* module, DEFORM simulation is to be performed. After completion of simulation, the results obtained may be displayed in *Post Processor* module in both graphical and data forms, in Figure 3, there has been given shape of a workpiece model with generated mesh of finite elements. After starting simulation process, numerical calculation for the stroke per step *0.1 mm*, are made, at which each tenth step is kept within data base. When the parameters of the finite element mesh reach their critical values, an automatic *remeshing* is done. For the given simulation example, *remeshing* was done 10 times in the whole simulation, and the process was finished at 16th step. A final look of the *model* workpiece in the 16th step with the finite element mesh is given in Figure 4. To compare data, it is necessary to input info sub-module *Point Tracking*, the adapted node points r_{p0} and z_{p0}. After inputting coordinates of node points for initial workpiece, generate coordinate of node points in the 16th or final step. Exporting data, node coordinate points at the end of deformation process in 16th phase r_{p16} and z_{p16} in *data* form are obtained. Based on the obtained node point coordinates at the end of deformation process, data processing is done by means of a programme made in the software package MATLAB. Input data are: node point coordinates at the beginning r_{p0} and z_{p0} and at the end of deformation process r_{p16} and z_{p16}. Dislocations of points are given by the following expression:

$$\left.\begin{aligned} u_{r16} &= r_{p16} - r_{p0} \\ u_{z16} &= z_{p16} - z_{p0} \end{aligned}\right\} \qquad (1)$$

Table 2: Insert object

			X mm	Y mm	R mm
Name: Upper die ☑Rigid	Geometry	1	0	45.87	0
		2	9.300732	45.87	1
		3	10	35.87	1
		4	19.300732	35.87	1
		5	20	25.87	1
		6	35	25.87	0
		7	35	45.87	0
	Movement			Speed constant 2 mm/s	
Name: Lower die ☑Rigid	Geometry		X mm	Y mm	R mm
		1	35	0	0
		2	35	10	0
		3	20	10	1
		4	19.300732	0	1
		5	10	0	1
		6	9.7202	3	1
		7	0	3	0
Name: Workpiece ☑Plastic	Geometry		X mm	Y mm	R mm
		1	0	3	0
		2	16.78	3	0
		3	16.78	35.87	0
		4	0	35.87	0
	Mesh	Number of Elements		1000	

Table 3: Inter-Object

Upper die - workpiece	Relation	☑Master - Slave
	Friction Type	Shear
	Friction Value	0.114
Lower die - workpiece	Relation	☑Master - Slave
	Friction Type	Shear
	Friction Value	0.114

Based on displacement of points u_{r16} and u_{z16} partial displacement derivation per radius and height are calculated: $\partial u_r/\partial r$, $\partial u_r/\partial z$, $\partial u_z/\partial z$ and $\partial u_z/\partial r$ in all 16 steps. Using such obtained partial derivations it is possible to determine components of small strains [6, 7, 8, 9, 10]. By applying a relation between relative strain and logarithmic strain (2), values of logarithmic strains are obtained to be comparable with numerical values of FEM simulation.

$$\varphi = \ln(1+\varepsilon) \qquad (2)$$

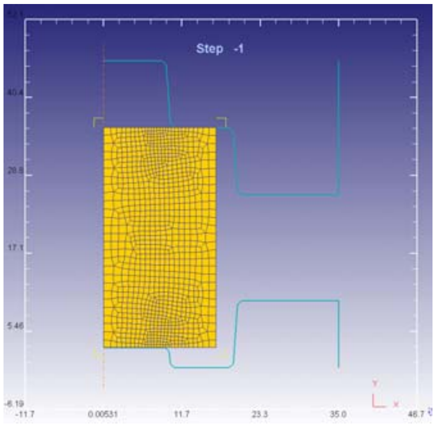

Fig. 3 - The initial workpiece model with generated mesh

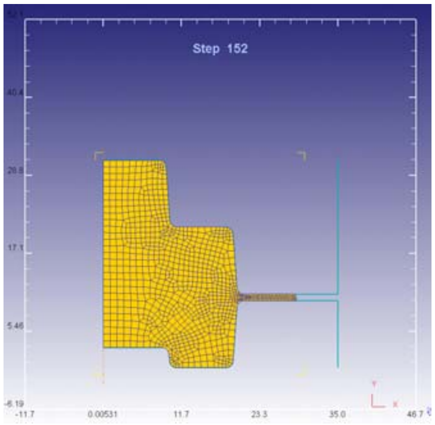

Fig. 4 - Workpiece model at the end of the process

The values of effective logarithmic strain are given in the form of a three-dimensional diagram in Figure 5.

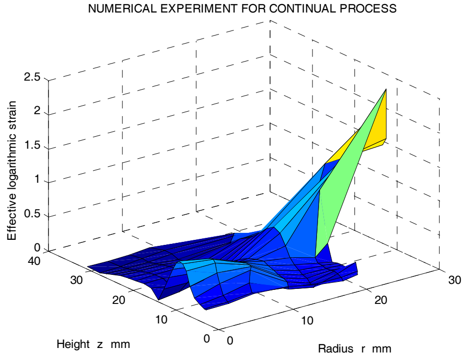

Fig. 5 - Effective logarithmic strain φ_e

3.2. Numerical experiment per steps

The data obtained by numerical simulation, due to a great number of possibilities DEFORM-2D software package possesses, relate to a complete bulk forming process may be reached by means of the generated data base. When numerical experiment in phases is concerned, the idea was to determine strain in each step of numerical experiment. To determine the values wanted, we get the needed data on radial and axial coordinates, adopted node points in each step separately out of DEFORM sub-Tracking-Point- module.

Based on the obtained node point coordinates per steps, data processing is made by means of a programme made in the software package MATLAB. Input data are: node point coordinates at the beginning r_{p0} and z_{p0} and per steps: r_{pi} and z_{pi}, $i = 1,2,...,16$. Displacements of points are given by the following expressions [11].

$$\left.\begin{aligned} u_{ri} &= r_{pi} - r_{pi-1} \\ u_{zi} &= z_{pi} - z_{pi-1} \end{aligned}\right\} \; i = 1,2,...,16 \tag{3}$$

Based on displacements of points u_{ri} and u_{zi}, $i = 1,2,...,16$ partial derivations of displacements for radius and height: $\partial u_r/\partial r$, $\partial u_r/\partial z$, $\partial u_z/\partial z$ and $\partial u_z/\partial r$ in each step are calculated. Out of these partial derivations, relative strains are obtained, where, by using (2) the logarithmic strains: φ_{ri} - radial, φ_{zi} - axial, $\varphi_{\theta i}$ - circular, γ_{rzi} - shear and φ_{ei} - effective, where: $i = 1,2,...,16$ at the end of each step of bulk forming process are obtained. Logarithmic strains values in the 16th step (at the end of bulk forming process) are calculated on the base of next equations:

$$\varphi_r = \sum_{i=1}^{16} \varphi_{ri} \,,\; \varphi_z = \sum_{i=1}^{16} \varphi_{zi} \,,\; \varphi_\theta = \sum_{i=1}^{16} \varphi_{\theta i} \,,\; \gamma_{rz} = \sum_{i=1}^{16} \gamma_{rzi} \;\text{ and }\; \varphi_e = \sum_{i=1}^{16} \varphi_{ei} \tag{4}$$

Effective logarithmic strain values obtained per steps are given in the form of a three dimensional diagram in Figure 6 [11].

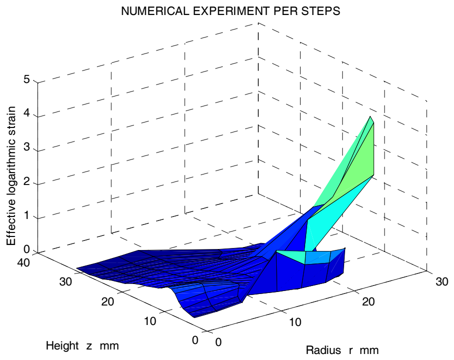

Fig. 6 - Effective logarithmic strain φ_e

3.3. DEFORM results

An important result that can be achieved by numerical simulation, apart from geometry change and dislocation of the points observed, is also strain distribution in each point of plane and axial-symmetrical deformations. DEFORM-2D software package has an advantage if compared to other software packages in the field of bulk forming, so it gives strain values in each step for the observed node points for the adopted node points r_{p0} and z_{p0} at the beginning of the process in non-deformed state.

To be able to compare data, it is necessary to input into sub-module *Point Tracking*, in advance, adopted node points r_{p0} and z_{p0}. After inputting node point coordinates in non-deformed state, node point displacement in each phase are generated and logarithmic strain values from the beginning to the end of deformation process are obtained in them. By *Sort by Step* order values of all parameters calculated by DEFORM for all adopted node points in advance are obtained [11].

Effective logarithmic strain component values obtained by DEFORM results, are given in the form of three-dimensional diagram in Figure 7 [11].

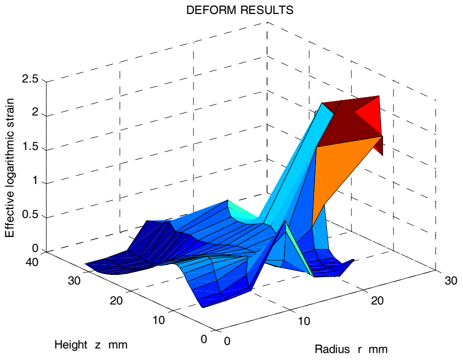

Fig.7 - Effective logarithmic strain φ_e

4. ANALYSIS AND COMPARISON OF THE OBTAINED RESULTS

On the base of previously given space diagram and deformation state parameter values derived from numerical experiment, an analysis and comparison of the obtained data have been made. For complete analysis and comparison, a programme was made in MATLAB that gives average diagrams in a meridian plane of a workpiece model for any cross-section. The cross-section P-P (Figure 8.) corresponds to a grade plane of a workpiece and of height value $z = 10.5$ *mm* [11].

Two-dimension logarithmic strain diagrams for all three stages research approaches are given in meridian plane of workpiece model for a characteristic cross-section P-P in Figure 9. to Figure 13.

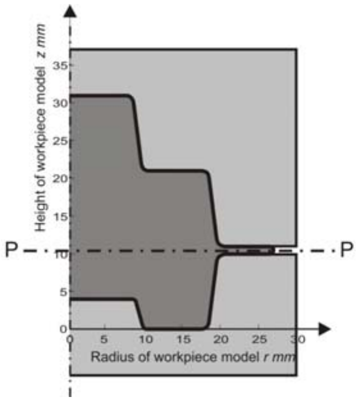

Fig. 8 - Characteristic cross-section in meridian plane for a half of axial-symmetrical workpiece

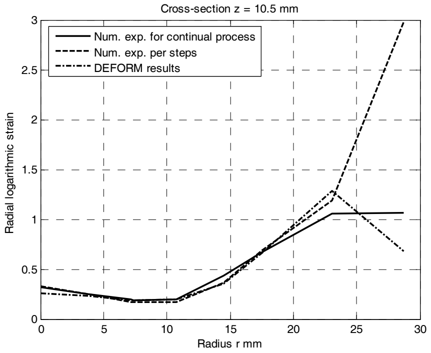

Fig.9 - Radial logarithmic strain φ_r for cross-section P-P

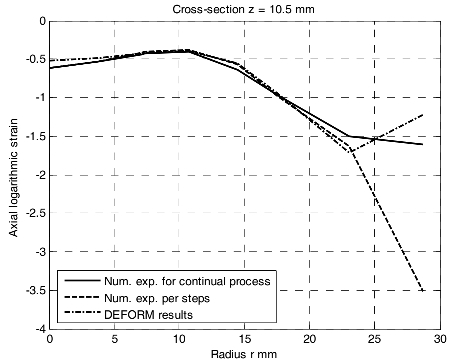

Fig.10 - Axial logarithmic strain φ_z for cross-section P-P

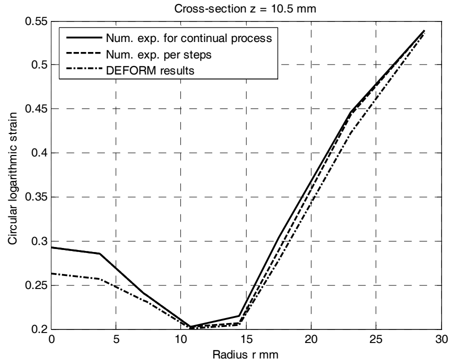

Fig. 11 - Circular logarithmic strain φ_θ for cross-section P-P

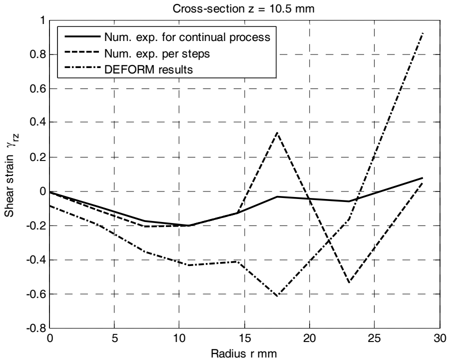

Fig.12 - Shear strain γ_{rz} for P-P cross-section

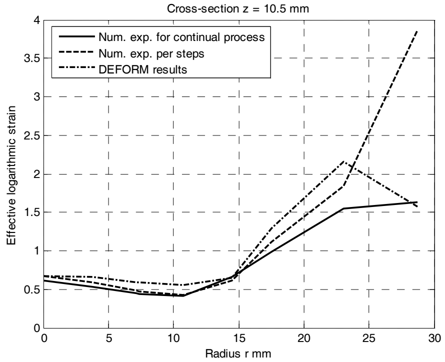

Fig. 13 - Effective logarithmic strain φ_e for P-P cross-section

At all the three research stages, for all strains, two zone values are clearly distinguished: flash zone and zone corresponding to inner die part (Figure 5. to Figure 7.).

Comparing logarithmic strains component, shear strains and effective logarithmic strains, at all the three research approaches of stepped convex die shape, it is evident that components are of similar change character and the obtained values are of the same value order. Maximum values are obtained in the flash zone with all the three research approaches, i.e. when high strains values are in question, it is evident the logarithmic strain values calculated per steps in the flash zone are higher than the ones calculated for process continuity. Deviations are evident with DEFORM results where extreme values appear at flash zone.

5. CONCLUSION

Strain state determination of concave-convex step workpiece models at bulk forming in open dies is elaborated in the paper. Research process was carried out in three stages: numerical experiment for continual process, numerical experiment per steps and DEFORM results.
Comparison and analysis of the obtained results are obtained, and conclusions are the following:

- Differences obtained by using a particular research approach relate mostly to the way of determining parameters. At numerical experiment for continual process, strains were determined by model of small strains for the whole forming process. At numerical experiment per steps, strains are determined by displacement per steps, where total strain is obtained as a sum of strains obtained for some steps. At DEFORM results, strains are determined by mathematical apparatus using DEFORM-2D.
- At discretization process per steps of bulk forming in open dies and determination of strain parameters in all phases, a more successful modelling of real process is achieved.

A strong expansion of computer engineering and software nowadays make it possible for body discretization and process discretization to be a way to greater accuracy in engineering researches.

6. REFERENCES

[1] DEFORM Mannual - SFTC.
[2] Janjić M., Vukčević M., Šibalić N.: Strain Modeling at Axi-Symmetrical Deformation in Open Die. 11th International Research/Expert Conference TMT 2007, Hammamet, Tunisia, 2007.
[3] Janjić M., Domazetović V., Savićević S., Vukčević M.: Određivanje krivih ojačavanja i njihov izbor u numeričkim simulacijama. XXXI JUPITER Konferencija, Zlatibor, 2005.
[4] Janjić M.: Istraživanje naponsko deformacionih parametara u procesima zapreminskog deformisanja - Doktorska disertacija. Mašinski fakultet, Podgorica, 2005.
[5] Janjić M., Vukčević M., Domazetović V.: Primjena metode diskretizacije u deformacionoj analizi odpreska. XXIV JUPITER Konferencija, Zlatibor, 1998.
[6] Musafija B.: Primijenjena teorija plastičnosti, Univerzitet u Sarajevu, Sarajevo, 1973.
[7] Musafija B.: Obrada metala plastičnom deformacijom. Svjetlost, Sarajevo, 1972.
[8] Plančak M.: Naponsko-deformaciono stanje u procesima hladnog istiskivanja čelika. Univerzitet u Novom Sadu, Fakultet Tehničkih Nauka, Novi Sad, 1984.
[9] Vukčević M., Janjić M., Šibalić N.: Deformation Modeling at Bulk Process Forming. 11th International Research/Expert Conference TMT 2007, Hammamet, Tunisia, 2007.
[10] Vukčević M., Janjić M., Domazetović V.: Uticaj gometrijskih parametara pri deformisanju osnosimetričnih uzoraka u otvorenim kalupima. XXVI JUPITER Konferencija, Beograd, 2000.
[11] Šibalić N.: Modeliranje procesa zapreminskog deformisanja korišćenjem metoda fizičke diskretizacije i numeričke simulacije - Magistarski rad. Mašinski fakultet, Podgorica, 2007.

FEM SIMULACIJE DEFORMACIONOG STANJA AKSIJALNO – SIMETRIČNIH ELEMENATA

Šibalić Nikola, Vukčević Milan, Savićević Sreten, Janjić Mileta

Univerzitet u Crnoj Gori,Mašinski fakultet u Podgorici, Crna Gora

REZIME

Tehnički i softverski razvoj omogućio je brzo i lako simuliranje kao i relativno lako određivanje potrebnih parametara plastično deformisani tela. Na taj način, proizvodnja je simulirana i poboljšana, tako da je omogućen brz razvoj novih proizvoda. Da bi se obezbedili relevantni podaci neophodno je spojiti tri metode: numeričku, teorijsku i eksperimentalnu. Pošto je poznavanje deformacionog stanja od izuzetne važnosti u procesima deformisanja, potrebni parametri za zapreminsko deformisanje su određeni numerički u slučaju osno – simetričnog radnog komada. Simulacija je urađena u DEFORM-2D softveru koji se bazira na metodi konačnih elemenata. Alat je u gornjem delu konveksan a u donjem konkavan, dimenzije alata i radnog komada su optimizovane. Simulacijom procesa moguće je izvesti diskretizaciju deformacije i odrediti parametre deformacije. Da bi se mogle određivati vrednosti parametara procesa kao i porediti dobijeni rezultati, praćene su određene tačke duž osnog preseka modela u eksperimentu i simulaciji. Numerički deo se sastoji od tri faze. U prvoj fazi je određivano pomeranje posmatranih tačaka na kraju procesa i određivane su deformacije. U drugoj fazi je određivano pomeranje posmatranih tačaka po fazama i deformacije za njih. Deformacije na kraju procesa su jednake sumi deformacija po fazama. Treća faza sadrži određivanje deformacija pomoću DEFORM-2D softvera. Zahtevani parametri su grafički interpretirani u radu. Takođe je prikazana i analiza i poređenje rezultata.

Ključne reči: d*eformacija, numerički eksperiment, simulacija, FEM, DEFORM-2D*