12th International Research/Expert Conference
"Trends in the Development of Machinery and Associated Technology"
TMT 2008, Istanbul, Turkey, 26-30 August, 2008

STRESS STATE MKE SIMULATIONS OF AXIS-SYMMETRICAL ELEMENT

Ph.D. Milan Vukčević
Ph.D. Mileta Janjić
M.Sc. Nikola Šibalić
University of Montenegro
Faculty of Mechanical Engineering
Cetinjski put bb, 81000 Podgorica
Montenegro

ABSTRACT

The development of computing technique and software has enabled a quick and easy simulation as well as a relatively simple deformination of stress and strain parametres 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 nee to connect three methods: numerical, theoretical and experimental.

As it is very important to know stress state parameters in deformation processes, a numerical experiment for axis-symmetrical elements 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 method of finite elements. An asymmetrical graded die shape is adopted being convex in its upper part and concave in its lower part, die and working-piece model dimensions are optimized. Simulating the process, it is possible to perform discretization per deformation and determine stress parameters in all phases. In the paper, node points arranged along the meridal section of working-piece model have been adopted, and will be followed in the course of experiment and their stress values will be determined. Numerical experiment consits of three stages.

In the first stage, displacement points at the end of deformation proces for the observed node points are determined, deformation-kinematic parameters are obtained, then by the method of visioplasticity, parameters stress are determined.

In the second stage, based on node point displacement and summing per phases, deformation-kinematic parameters are obtained, then by the method of visioplasticity, stress parameters are determined.

In the third stage, stress out of DEFORM-2D software package in the observed points are obtained.

The required parameters are graphically interpreted in the paper. Analysis and comparison of the results obtained are made as well.

Keywords: Stress, Deformation, Numerical experiment, Simulation, FEM, DEFORM-2D, Visioplasticity.

1. INTRODUCTION

Stress state parametres are great importance for a successful projection of any technological strain procedure. Due to a rapid development of computer engineering and software, a lot of commercial programme packages in the field of deformation have been developed in the recent years. One of the most frequently used programmes for deformation process simulation is the DEFORM package consisting of several modules: DEFORM-2D, DEFORM-3D, DEFORM-PC, DEFORM-PC Pro and DEFORM-HT. The DEFORM-2D package is based on parametre procecess modelling and finite element methods (FEM) are used in this paper. The DEFORM-2D Package is intended to plane deformation.

2. INPUT PARAMETERS

To perform a successful numerical simulation of deformation process, it is necessary to adopt some input parameters and a proper die shape.The data relating to *AlMgSi0,5*, aluminium alloy being pressed at hot forming temperature $T= 440\ [^oC]$ are used in numerical experiment. An asymmetrical graded tool shape (Fig. 1.), consisting of two dies, upper and lower one, is adopted. The upper part is convex and consists of two height grades, wheras lower one is concave, also consisting of two, height grades, one of them being omitted. Also, the following data are adopted in the paper:

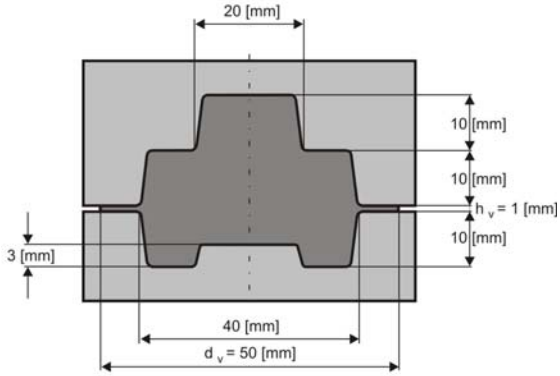

Figure 1. Working-piece model within die

- Deformation is obtained at small constant deformation speed, $v=2\ [mm/s]$.
- Strain hardening curve parametres are $c=30.34434$ and $n=0.097808$ for *AlMgSi0.5* aluminium alloy and temperature is $T=440\ [^oC]$.
- Friction factor is $m=0.114$.

The working-piece model is cylindrical, with diameter of $d_0=33.56\ [mm]$ and height $h_0=32.87\ [mm]$. Coordinate point whose displacement will be followed in numerical experiment and whose stress parameters will be determined are shown in Fig. 2. An adopted net of a half of axi-symmetrical working-piece model has 154 node points.

3. NUMERICAL SIMULATION

As DEFORM-2D consists of three main modules: *Pre Processor, Simulator* and *Post Processor*, it is necessary for the adopted parameters to input them into *Pre Processor* module and adjust other relevant parametrers Table 1. Before starting numerical simulation, it is necessary to check the system by using *Check* option, namely to check whether the process is really established and whether all the parameters are properly set. After inputting data and forming a mesh of finite elements, data base is generated in sub-module *Database Generation* for initial step marked -1. Generating data base, all the needed conditions for performing deformation analysis are acquired, namely by entering *Simulator* module, DEFORM simulation is carried out.

Table 1. Pre Processor

Main	Units	☑SI			
	Geometry	☑Axisymmetric			
	Type	Incremental			
	Mode	Deformation			
Step	Number of Simulation Steps	1000			
	Step Increment to Save	10			
	Primary Die	1 - Gornji kalup			
	With Equal Die Displacment	0.1 [mm]			
Stop	Primary Die Displacement	0	14.87 [mm]		
Name: Upper die ☑Rigid	Geometry		X [mm]	Y [mm]	R [mm]
		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: Working-piece ☑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		
Upper die - working-piece	Relation	☑Master - Slave			
	Friction Type	Shear			
	Friction Value	0.114			
Lower die - working-piece	Relation	☑Master - Slave			
	Friction Type	Shear			
	Friction Value	0.114			

The Simulation done, the results obtained can be interpreted in *Post Processor* module in both graph and data forms. In Fig. 2. there is the look of the working-piece model with the generated mesh of finite elements in seizure with dies. Numerical calculation are done for stroke value per step of *0.1 [mm]*. An automatic remeshing is done when parametrers of finite element mesh reach their critical values. For the given simulation example, remeshing is carried out 10 times, and the process ended in the 16th phase. The final look of the working-piece in the 16th phase with finite element mesh in seizure with dies is given in Fig. 3. To compare data, it is necessary to input in advance the adopted node points r_{p0} and z_{p0}, into *Point Tracking* sub-module, and their arrangement is given in Fig. 2. After inputting coordinates of node points in non-deformed state into *Point Tracking* sub-module, coordinate of node points are generated either in the 16th or final phase, and their arrangement is given in Fig. 3. Exporting data, coordinates of node points at the end of deformation process in 16th phase r_{p16} and z_{p16} in data form are obtained.

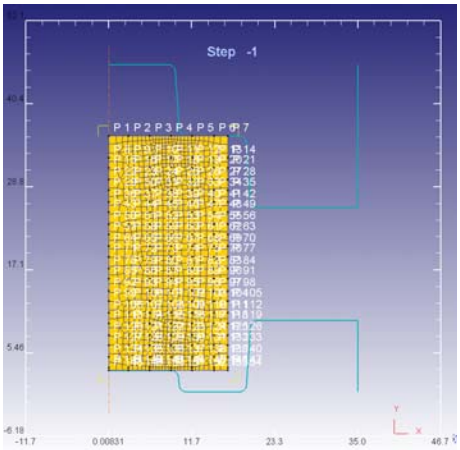

Figure 2. Adopted arrangement of node points in non-deformed state

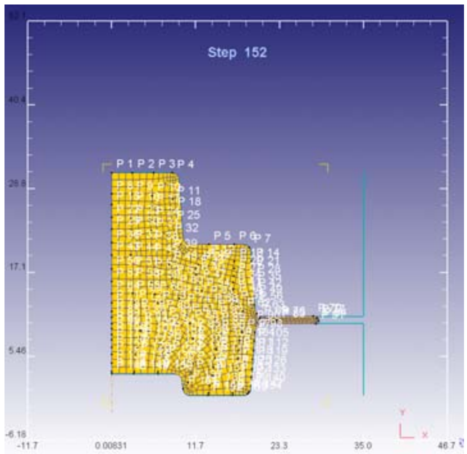

Figure 3. Point Arrangement in 16th phase obtained by DEFORM simulation

3.1. Numerical experiment for process continuity

Determining deformation and kinematic parameters is done by virtue of the obtained coordinates of node points at the end of deformation process, namely point dislocation displacemend. Based on the numerical simulation data, deformation components and deformation speed [3] are obtained.

Stress is determined by visioplasticity method. Data are processed in MATLAB. Input data are initial node point coordinates r_{p0} and z_{p0} and those at the end of deformation process r_{pk} and z_{pk}, $k=16$, strengthening curve parametres c and n, as well as the results obtained out of deformation φ_r, φ_z, φ_θ, γ_{rz} and φ_e and kinematic analyses $\dot{\varepsilon}_r$, $\dot{\varepsilon}_z$, $\dot{\varepsilon}_\theta$, $\dot{\gamma}_{rz}$ and $\dot{\varepsilon}_e$ [3].

The method is based on obtaining axial σ_z stress component by solving the basic visioplasticity equation [1], where the main problem appers to be determining integration constant C. Only points to determine axial component value are stress points for maximum radius value at wreath level. These values are determined out of the condition where radial stress component in those points equals zero:σ_r=0. Other deformation kinematic parameters are known, thus effective stress is determined by corresponding hardening curves for effective deformation value.

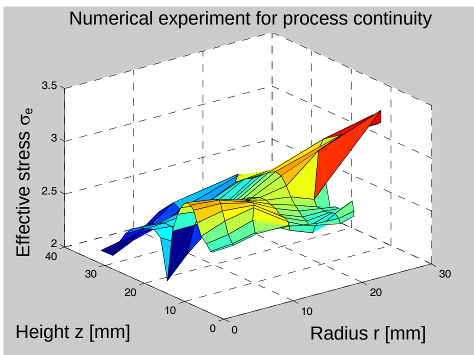

Figure 4. Effective stress σ_e [daN/mm^2]

Effective stress values at the end of deformation process in the observed points of meridial model cross-section of a working-piece are in the form of three-dimension diagram in Fig. 4.

3.2. Numerical experiment per steps

At numerical experiment per steps, deformations in each step are determined, where final deformation value and speed are obtained by known procedures and methods [3].

Stress is determined analogously using visioplasticity method. Input data are: initial node point coordinates r_{p0} and z_{p0} and those at the end of deformation process r_{pk} and z_{pk}, $k=16$, hardening curve parameters c and n, and the results obtained from both deformation φ_r, φ_z, φ_θ, γ_{rz} and φ_e and kinematic analyses $\dot{\varepsilon}_r$, $\dot{\varepsilon}_z$, $\dot{\varepsilon}_\theta$, $\dot{\gamma}_{rz}$ and $\dot{\varepsilon}_e$ [3].

Final values of effective stress are given in the form of three-dimension diagram in Fig. 5.

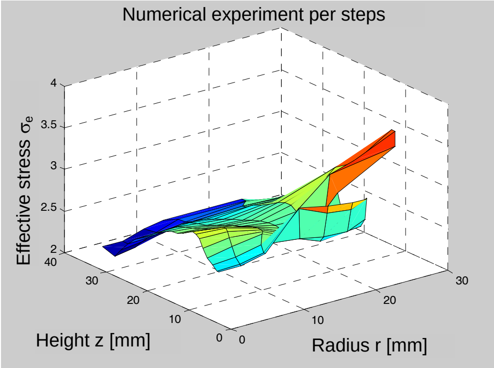

Figure 5. Effective stress σ_e [daN/mm^2]

3.3. DEFORM results

Stress chang values per each step of the observed process for adopted node points are obtained from DEFORM-2D software package directly.

The values of this effective stress obtained at the end of deformation process are given in Fig. 6.

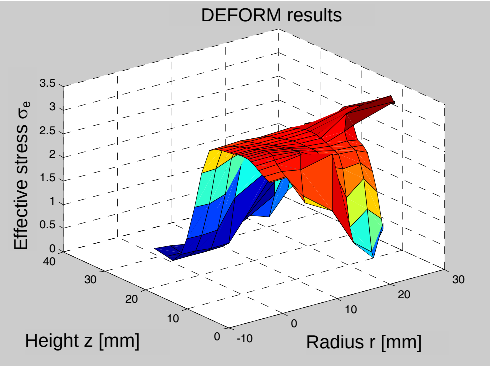

Figure 6. Effective stress σ_e [daN/mm^2]

4. CONCLUDING CONSIDERATION

Stress state differences obtained by using a special research procedure are primarily related to the way of deformation state parameter determination. At numerical experiment for continuity process, deformations are determined by model of small deformations for the whole deformation process. At numerical experiment per steps, deformation is determined by virtue of displacement per steps, where total deformation is obtained as a sum of deformations per steps. At DEFORM results, deformations are determined by means of mathematical apparatus using DEFORM-2D. Based on the stress results obtained and presented in this paper and analysis done, at may be concluded that, within numerical simulations of deformation processes for the stated conditions, numerical experiment per deformation steps in much more appropriate thus discretization per steps is necessary in research process. A great expansion of computer engineering and software nowadays has made it possible for engineering research, body discretization and process discretization to be a good way to increasing accuracy.

5. REFERENCES

[1] Janjić M.: Istraživanje naponsko deformacionih parametara u procesima zapreminskog deformisanja - Doktorska disertacija. Mašinski fakultet, Podgorica, 2005.

[2] Vukčević M., Janjić M., Šibalić N.: Numerical Simulation in bulk forming process. Journal for Technology of Plasticity, Vol. 32 (2007), Number 1-2.

[3] Šibalić N.: Modeliranje procesa zapreminskog deformisanja korišćenjem metoda fizičke diskretizacije i numeričke simulacije - Magistarski rad. Mašinski fakultet, Podgorica, 2007.

[4] DEFORM Mannual - SFTC.