Journal of Trends in the Development of Machinery and Associated Technology
Vol. 20, No. 1, 2016, ISSN 2303-4009 (online), p.p. 21-24

FEM SIMULATION OF ALUMINIUM ALLOY ON THE HIGH TEMPERATURE

Ph.D. Milanko Damjanovic
Ph.D. Milan Vukcevic
Ph.D. Nikola Sibalic
University of Montenegro
Faculty of Mechanical Engineering
Dzordza Vasingtona bb, 81000 Podgorica
Montenegro

ABSTRACT
Investigation of strain state of metal forming in open dies has been presented in the paper. Logaritmic strain tensor components are determined by meridial point cross-section per deformation phase displacements, where a total strain is calculated as a sum of strains in deformation phases. For FEM simulation, DEFORM 2D softwer package is used. FEM simulation consists of two approach.
Keywords: FEM simulation, High temperature, DEFORM 2D

1. INTRODUCTION
As a result of a rapid development of computer engineering in the field of metal forming a number of different numerical methods has been developed and used so far. Finite Elements Method - FEM has been used as the most powerful numerical method. Based on this method, several commercial software packages for numerical process simulations have been developed. Among them, one of the most famous software package is DEFORM, produced by *Scientific Forming Technologies Corporation* (SFTC) which has been used in this paper [2, 3, 4, 5].

2. INPUT PARAMETERS
Metal forming process in open dies includes a wide class of problems, both from the aspect of geometry and technological condition. The elements given in Figure 1. has been analysed in the paper.

- Experimental material is aluminium alloy *AlMgSi*0.5.
- Deformation is realized at low constant deformation speed, $v = 2$ *mm*/*s*.
- Hardening curve parametres are c = 30.34434 an n = 0.097808 for *AlMgSi*0.5 aluminium alloy and high temperature $T = 440\ ^oC$.
- Friction factor is $m = 0.114$.
- Tool is of stepped concave shape (Figure 1.). It consists of two pieces, upper and lower ones. The upper die piece consists of two height degrees where one is with degree, whereas lower one consists of one height degree.

Workpiece are cyllindrical, of diameter $d_0 = 33.56$ *mm*. The height h_0 is determined out of the workpiece constant bulk before and after pressing process for adopted die dimensions given in Figure 1. and it is $h_0 = 29.58$ *mm*. The adopted mesh of the half of the axi-symmetrical workpiece is give in Figure 2. Total node point number is 140. Point coordinates, whose displacement will be followed in numerical experiment and whose strain state parameters will be determined, are given in Figure 3. [1].

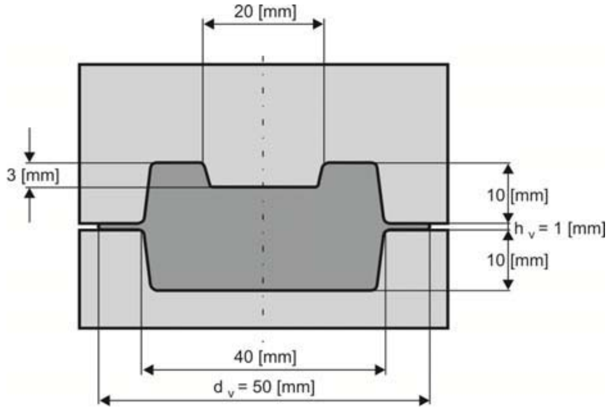

Figure 1. Workpiece with in open die [1]

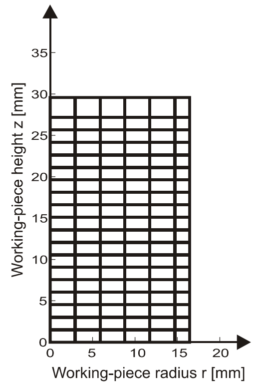

Figure 2. Adopted mesh of the halt of axi-symmetrical workpiece [1]

3. FEM SIMULATION

For the adopted tool geometry (Figure 1.) and adopted workpiece geometry (Figure 2.), as well as for known friction and hardening curve factors, FEM simulation has been carried out, where the data input into DEFORM's module *Pre Processor*, are given in Table 1.

Table 1. Input data of DEFORM simulation for concave die shape [1]

<table>
<tr><td rowspan="2">Simulation Parameters</td><td colspan="3">Units UNIT</td><td colspan="2">☑SI</td></tr>
<tr><td colspan="3">Geometry GEOTYP</td><td colspan="2">☑Axisymmetric</td></tr>
<tr><td rowspan="5">Step Controls</td><td colspan="3">Number of simulation steps</td><td colspan="2">NSTEP=1000</td></tr>
<tr><td colspan="3">Step increment to save</td><td colspan="2">STPINC=10</td></tr>
<tr><td colspan="3">Primary die</td><td colspan="2">PDIE(1)=1</td></tr>
<tr><td colspan="3">Steps by</td><td colspan="2">☑Stroke</td></tr>
<tr><td colspan="3">Stroke per step</td><td colspan="2">DSMAX=0.1 mm</td></tr>
<tr><td>Stopping Controls</td><td colspan="3">Primary die displacement</td><td colspan="2">SMAX=0, 11.58 mm</td></tr>
<tr><td rowspan="10">Name: Upper die
☑Rigid</td><td rowspan="8">Geometry</td><td></td><td>X mm</td><td>Y mm</td><td>R mm</td></tr>
<tr><td>1</td><td>0</td><td>29.58</td><td>0</td></tr>
<tr><td>2</td><td>9.7202</td><td>29.58</td><td>1</td></tr>
<tr><td>3</td><td>10</td><td>32.58</td><td>1</td></tr>
<tr><td>4</td><td>19.300732</td><td>32.58</td><td>1</td></tr>
<tr><td>5</td><td>20</td><td>22.58</td><td>1</td></tr>
<tr><td>6</td><td>35</td><td>22.58</td><td>0</td></tr>
<tr><td>7</td><td>35</td><td>32.58</td><td>0</td></tr>
<tr><td colspan="3" rowspan="2">Movement controls</td><td colspan="2">Speed 2 mm /s</td></tr>
<tr><td colspan="2">Angle -90°</td></tr>
<tr><td rowspan="6">Name: Lower die
☑Rigid</td><td rowspan="6">Geometry</td><td></td><td>X mm</td><td>Y mm</td><td>R mm</td></tr>
<tr><td>1</td><td>35</td><td>0</td><td>0</td></tr>
<tr><td>2</td><td>35</td><td>10</td><td>0</td></tr>
<tr><td>3</td><td>20</td><td>10</td><td>1</td></tr>
<tr><td>4</td><td>19.300732</td><td>0</td><td>1</td></tr>
<tr><td>5</td><td>0</td><td>0</td><td>0</td></tr>
<tr><td rowspan="6">Name: Workpiece
☑Plastic</td><td rowspan="5">Geometry</td><td></td><td>X mm</td><td>Y mm</td><td>R mm</td></tr>
<tr><td>1</td><td>0</td><td>0</td><td>0</td></tr>
<tr><td>2</td><td>16.78</td><td>0</td><td>0</td></tr>
<tr><td>3</td><td>16.78</td><td>29.58</td><td>0</td></tr>
<tr><td>4</td><td>0</td><td>29.58</td><td>0</td></tr>
<tr><td>Mesh</td><td colspan="3">Number of mesh elements</td><td>MGNELM=1000</td></tr>
<tr><td rowspan="3">Upper die -
Workpiece</td><td colspan="4">Contact relation CNTACT</td><td>☑Master-Slave</td></tr>
<tr><td colspan="4">Friction model FRCFAC</td><td>Shear</td></tr>
<tr><td colspan="4">Friction</td><td>FRCFAC=0.114</td></tr>
<tr><td rowspan="3">Lower die -
Workpiece</td><td colspan="4">Contact relation CNTACT</td><td>☑Master-Slave</td></tr>
<tr><td colspan="4">Friction model FRCFAC</td><td>Shear</td></tr>
<tr><td colspan="4">Friction</td><td>FRCFAC=0.114</td></tr>
</table>

Finite element mesh forming process and data base generation are valid for the initial step "-1", (Figure 3.), During deformation process at the high temperature 440 °C, remeshing was done four times, and the process ended in 12th phase. To compare the results, it is necessary to input the adopted node points r_{p0} and z_{p0}, into *Point Tracking* sub-module, and their distribution is given in Figure 3. After inputing node point coordinates of non-deformed state into *Point Tracking* sub-module, node point coordinates per phases are generated. The distribution of point in 12th or final phase is given in Figure 4. Using *Data Extract* order, node point coordinates per phases are also obtsined at the end of deformation process, namely in 12th phase r_{p12} and z_{p12}.

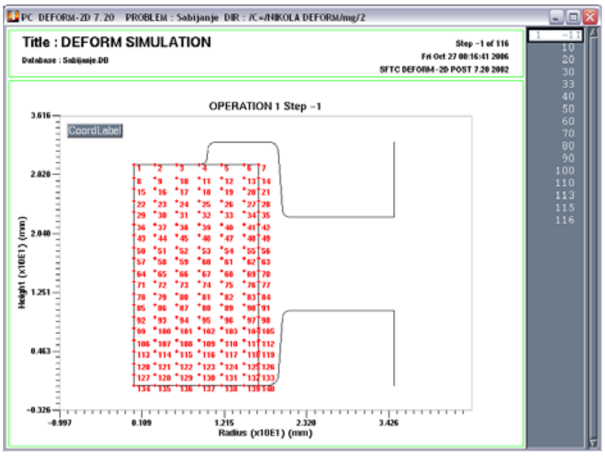

Figure 3. Adopted distribution of node point in non-deformed state [1]

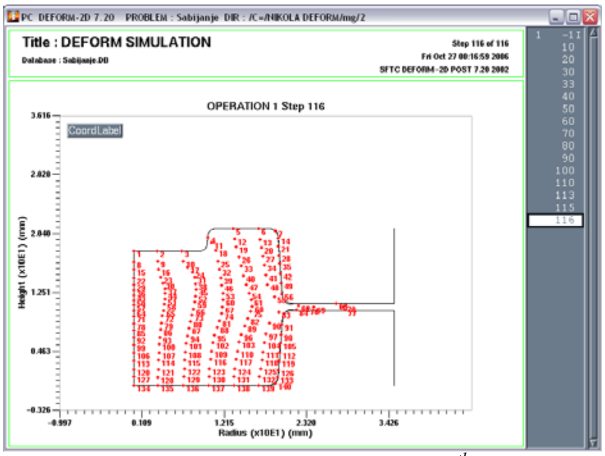

Figure 4. Point distribution in 12th phase obtained by DEFORM simulation [1]

Strain parameters have been determined by the obtained node points at the end of deformation process. Input data are point coordinates at the beginning r_{p0}, z_{p0} and at the end of deform process r_{p12}, z_{p12}, (Figure 3. and Figure 4.). Point displacements are expressed by:

$$\begin{aligned} u_{r12} &= r_{p12} - r_{p0} \\ u_{z12} &= z_{p12} - z_{p0} \end{aligned} . \qquad \text{... (1)}$$

Using displacements point u_{r12} and u_{z12}, partial displacement statements per radius and height are calculated: $\partial u_r/\partial r$, $\partial u_r/\partial z$, $\partial u_z/\partial z$ and $\partial u_z/\partial r$. Based on these statements, components of small deformations may be determined [3, 6]. Using relations (2) logarithmic strain values are obtained to be compared to the values of numerical FEM simulation.

$$\varphi = \ln(1+\varepsilon) . \qquad \text{... (2)}$$

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

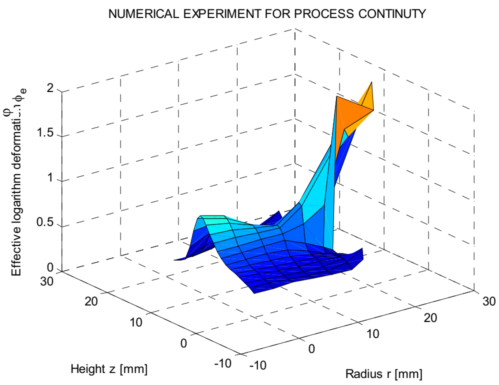

Figure 5. Effective logarithmic strain φ_e [1]

Strain values on the high temperature, in each phase of the observed process, for the adopted node points are obtained directly out of DEFORM-2D software package [7].
Values of these components at the end of deform process, effective logarithmic strains are given in Figure 6.

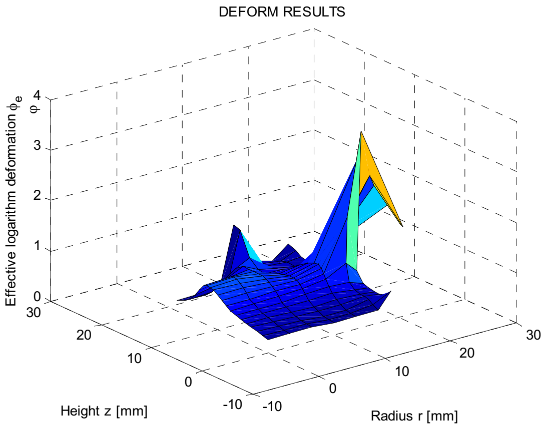

Figure 6. Effective logarithmic strain φ_e [1]

4. CONCLUSION

FEM simulations of deformation process on the high temparature, have been carried out in the paper, by using finite element method. Experimental material is aluminium alloy. The numerical experiments carried out, simulate the conditions of a real object. Strain parameters of a workpiece in open dies of axi-symmetrical elements have been determined. FEM simulation is carried out for a stepped concave die shape and it consists of two approach: numerical experiment for process continuty and DEFORM results.

5. REFERENCES

[1] Sibalic N.: Modeling of bulk metal forming processes using methods of physical discretization and numerical simulation - Master thesis, Faculty of mechanical Enginering, Podgorica, 2007.,
[2] DEFORM Mannual - SFTC,
[3] Musafija B.: Applied Plasticity Theory, University in Sarajevo, Sarajevo, 1973.,
[4] Musafija B.: Plastic deformation Metal Working. Svjetlost, Sarajevo, 1972.,
[5] Plancak M.: Strain-deformation state in processes of cold steel extrusion. Faculty of Engineering Sciences, Novi Sad, 1984.,
[6] Causević M.: Theory of Plastic Metal-Working. Svjetlost, Sarajevo, 1979.
[7] Vukcevic M., Sibalic N., Savicevic S., Janjic M.: FEM Simulation and Determination of Stress of the Alloy AlMgSi0.5 in Open Dies, International virtual journal for science, technics and innovations for the industry: Machines, Technologies, Materials - MTM, Year VII, Issue 10/2013, ISSN 1313-0226, p. 23-26.