XII International Conference on Computational Plasticity. Fundamentals and Applications
COMPLAS XII
E. Oñate, D.R.J. Owen, D. Peric and B. Suárez (Eds)

NUMERICAL SIMULATION OF SEVERE PLASTIC DEFORMATION DURING HIGH PRESSURE TORSION

P. VERLEYSEN*, F. VAN DEN ABEELE† AND J. DEGRIECK*

* Department of Materials Science and Engineering
Faculty of Engineering and Architecture
Ghent University
Technologiepark 903, 9052 Zwijnaarde (Ghent), Belgium
email: Patricia.Verleysen@UGent.be, http://www.ugent.be/ea/dmse/en

† Vikar R&D, Beukenpark 65, 9880 Aalter, Belgium
E-mail: filip@Vikar.be

Key words: Severe Plastic Deformation, High Pressure Torsion, Finite Elements, ALE, Remeshing.

Abstract. The principle of achieving high strength and superior properties in metal alloys through the application of severe plastic deformation has been exploited in the metal processing industry for many decades. The High Pressure Torsion (HPT) process is one of the most promising techniques which imposes very high strains to a bulk solid without introducing a significant change in sample dimensions. In this paper finite element simulations are presented. The simulations aim at in-depth understanding of the HPT process and the factors affecting this process. Above all, the extremely large deformations involved are challenging for the numerical algorithms used; traditional meshing techniques fail. In this contribution the use of advanced meshing and remeshing techniques implemented in the commercial finite element code ABAQUS is discussed. Results are presented of HPT simulations showing that the design and material of the mold, and the corresponding sample dimensions are the major factors affecting the HPT process.

1 INTRODUCTION

Nanocrystalline materials (NCM) represent a whole generation of solids with new atomic structures and properties by utilizing the atomic arrangements in the cores of defects such as grain boundaries, interphase boundaries or dislocations. Severe plastic deformation (SPD) achieves very fine crystals by the use of severe straining and deforming the material under high pressure to prevent the material from failing. SPD processing is defined in [1] as any method of metal forming under an extensive hydrostatic pressure that may be used to impart a very high strain to a bulk solid without the introduction of any significant change in the overall dimensions of the sample and having the ability to produce exceptional grain refinement. The most important methods to achieve SPD are Equal Channel Angular Pressing (ECAP) and high pressure torsion.

During a HPT process a disk-like specimen, with a height around 1mm and a diameter between 10 and 20mm, is compressed between two anvils. The pressure varies from 1GPa up to 10GPa. Once the pressure is applied, one anvil is rotated with respect to the other.

Due to friction in contact surfaces between the specimen and the anvils, the specimen is deformed by shear force. The main volume of the specimen is strained under hydrostatic compression because of the applied pressure and the pressure of the outer layers of the sample.

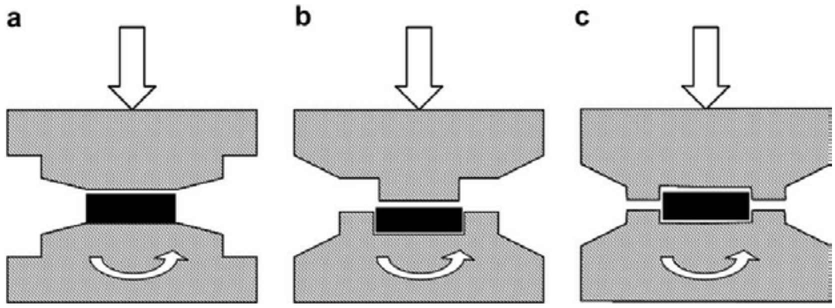

Figure 1: Unconstrained (a), constrained (b) and semi-constrained (c) HPT [1]

Two distinct types of high pressure torsion exist, each with its own features. The first type, see figure 1a [1], is called unconstrained. The sample is placed between two flat anvils during the test. This type has the advantage of a simple anvil design. The downside of this technique is the difficulty to accomplish a hydrostatic pressure throughout the specimen. When the pressure is applied, and the work piece is rotated, nothing obstructs the outward flow of material at the edges. As a result, the diameter of the sample increases while in the center the backpressure due to friction is large, and no outward flow is possible. As a result, the thickness varies as function of the radius, with inhomogeneous properties as a result.

The second type is represented in figure 1b and is called constrained. The sample is placed within the mold, which prevents outward flow completely. The hydrostatic pressure is constant throughout the specimen with more homogeneous properties after processing. The difficulty lies in the mold: designing the geometry and selection of appropriate materials is a difficult challenge. When the compression force is applied, both the specimen and the mold decrease in height. As a result, it is likely that the molds will come into contact and introduce high friction forces and wear rates in the contact surface.

A compromise is found when using the quasi-constrained geometry. The outward flow is partially prohibited by placing the specimen in depressions in the anvils. However, the height of the specimen is larger than the depth of the depressions. This will result in large friction forces at the edges, ensuring hydrostatic pressure in the specimen, and lower wear rates in the anvils. This is the kind of HPT found in most literature and discussed here.

The grain refinement and the corresponding enhancement of mechanical properties obtained by a HPT process are to a large extent dependent on the pressure and strain imposed to the material. The distribution of the strain and the pressure in a sample are determined by the geometry and material of both the mold and the sample, and the friction between mold and sample. Numerical simulations have been used to study the conditions of the HPT process.

In [2], the 3D deformation behaviour of a high density polyethylene (HDPE) sample during an unconstrained high pressure torsion process is simulated using MSC.Marc. The initial dimensions of the cylindrical sample are a diameter of 20 mm and a height 10 mm. The sample is meshed with 34 992 eight-node isoparametric hexahedral elements. The upper and lower anvils are assumed to be rigid bodies. A vertical displacement is imposed to the upper anvil in the compressive direction, and is maintained during torsion.

In [3], the DEFORM finite element code is used to perform isothermal simulations of the unconstrained HPT for pure copper. The initial dimensions of the copper work piece are 20 mm in diameter and 10 mm in height. The anvils are modelled as rigid bodies, and are assumed to be tied to the deforming sample. Kim [4,5] report isothermal finite element simulations of unconstrained HPT for pure copper using Abaqus. The cylindrical specimen with diameter 20 mm and height 0.7 mm is modelled using four-node generalised axisymmetric elements with twist (CGAX4). These elements allow for displacements in the circumferential direction that vary with radius and height.

In this paper, the implicit, commercial FE code Abaqus/standard is used to simulate HPT processes. Attention is focussed on meshing and remeshing techniques able to cope with the extremely large deformations and corresponding mesh distortion that occurs during severe plastic deformation.

2 FINITE ELEMENT MODEL

2.1 Reference model

For the finite element analyses, a disk shaped specimen with diameter D = 10 mm and thickness H = 1.0 mm is used. Semi-constrained conditions are adopted. A detail of the geometry of the upper part of the mold selected as the reference mold for this study is shown in Figure 2. The geometry is based on the molds used by CEIT (San Sebastian, Spain). In the finite element model only the lower mold and half of the specimen -along the thickness direction- is modeled, using an appropriate symmetry boundary condition.

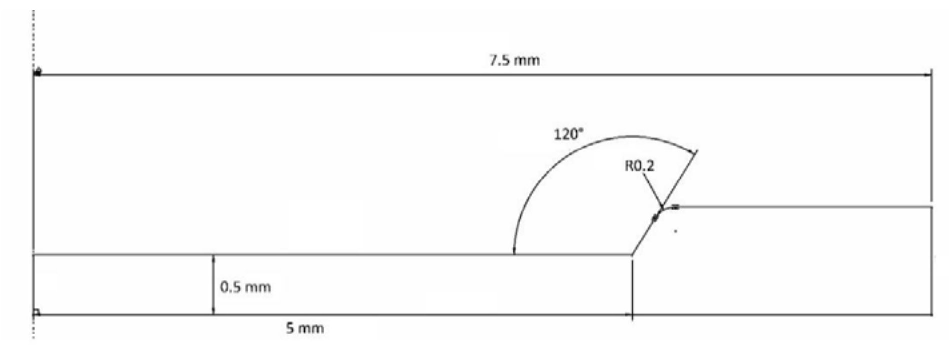

Figure 2: Geometry and dimensions of an axisymmetric section of the upper part (containing the sample cavity) of the die used as reference

For both the sample and the mold, a classical metal, elasto-plasticity model with isotropic hardening is used. The sample is modelled as soft steel, with a Young's modulus E = 210 GPa and a Poisson coefficient ν=0.3, the elasto-plastic properties are defined by a yield stress σ_y = 100 MPa, and a small-slope hardening curve that allows for severe plastic deformation (up to

500% strain). For the steel mold different yield stress levels are considered in the range of [500MPa;2500MPa] and again small-slope hardening upto 4% of plastic strain.

Coulomb frictional contact conditions (with a friction coefficient μ=0.1) are applied at the interface between the steel sample and the die. Initially, the sample is meshed with 50 x 10 CGAX4 elements, a mesh density suggested by [4]. As explained in section 2.2 for the final calculation a more advanced mesh strategy is adopted. The CGAX4 element is a four node bilinear axisymmetric quadrilateral element allowing rotation. It is useful for the analysis of structures that are axially symmetric, but can twist about their symmetric axis, as is the case in high pressure torsion processes. Indeed, the CGAX4 element allows displacements in the circumferential (θ) direction that vary with the radius and the height, but not with θ.

2.2 Enhanced mesh strategy

With the mesh described in section 2.1, in a first step, a vertical displacement u_y = 0.04 mm is imposed on the surface of the lower part of the die, to simulate compressive loading of the steel specimen. The deformation of the sample, and the corresponding compressive stresses, are shown on Figure 3. At (very) high levels of compressive strain (up to 3.5 GPa), however, the simulation seems to suffer from mesh distortion. The soft steel sample cannot cope with the extreme plastic deformation at the extremity where the sample is in contact with the mold, and convergence during a subsequent torsion loading step is unlikely. Therefore, to introduce more elements in the vicinity of the inclined shoulder, the outer edge of the die is meshed separately. A higher mesh density in this critical area of the master surface clearly improves the stability of the contact algorithm. Additionally, a biased mesh is used over the thickness of the sample, to anticipate to large deformations at the extremity of the sample. Next to that also the adaptive remeshing capabilities of Abaqus to maintain reasonable element shapes and aspect ratios whilst the shape of the sample is evolving are activated. In Figure 4, the initial simulation is compared with an enhanced mesh strategy, indicating that the latter solution can cope with extreme plastic deformation during the compression step.

During the torsion step, however, the optimized mesh strategy is insufficient to reach a deformation corresponding with at least 1 rotation. Also other techniques available in Abaqus, such as Eulerian meshing and mixed Eulerian-Lagrangian adaptive meshing approaches fail. Finally, the desired deformation is obtained by consecutive restart/remeshing steps. Although, the restart/remeshing procedure is demanding and oscillations are introduced in the results, the desired rotation can be imposed.

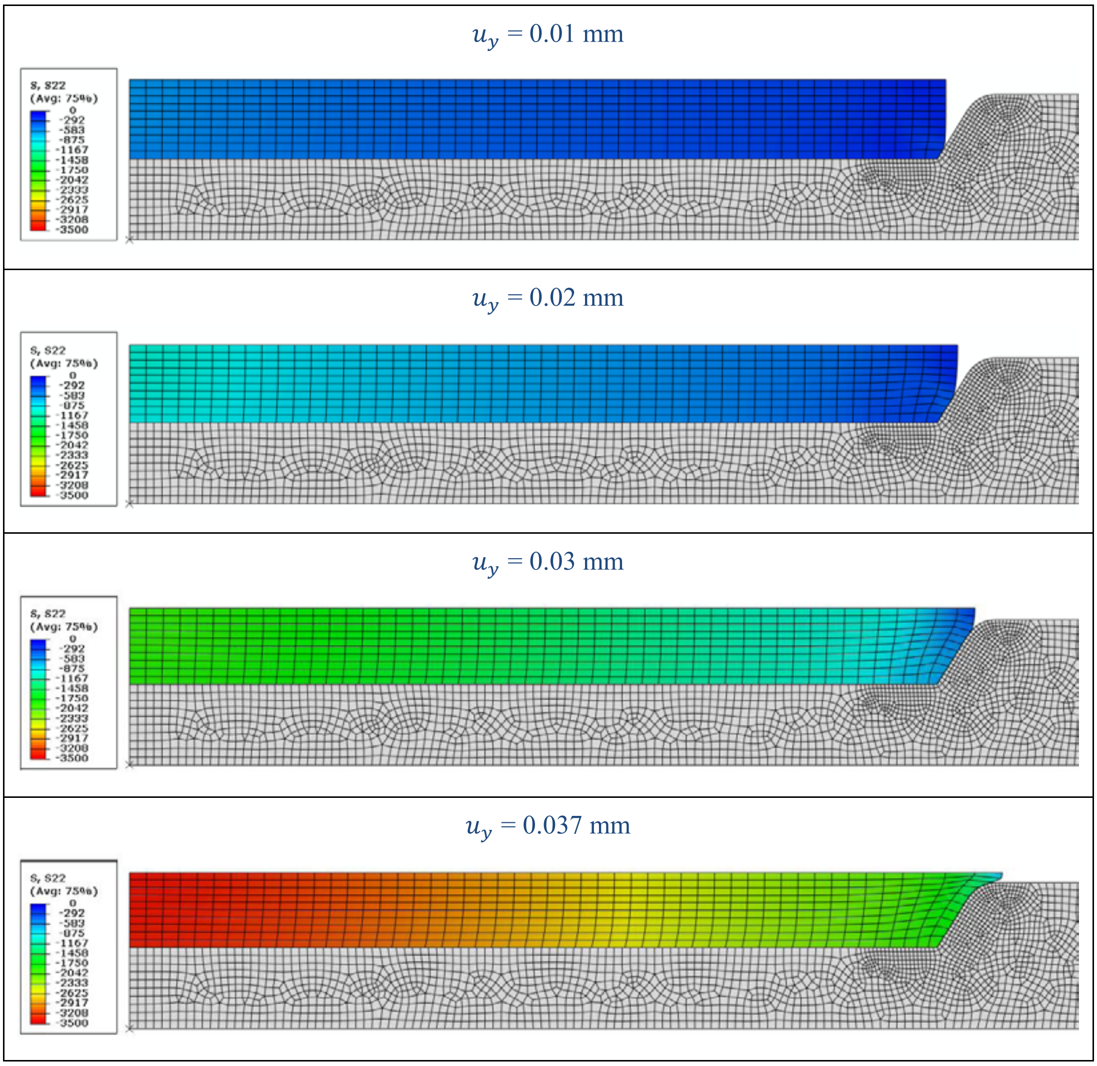

Figure 3: Mesh distortion during application of compressive load

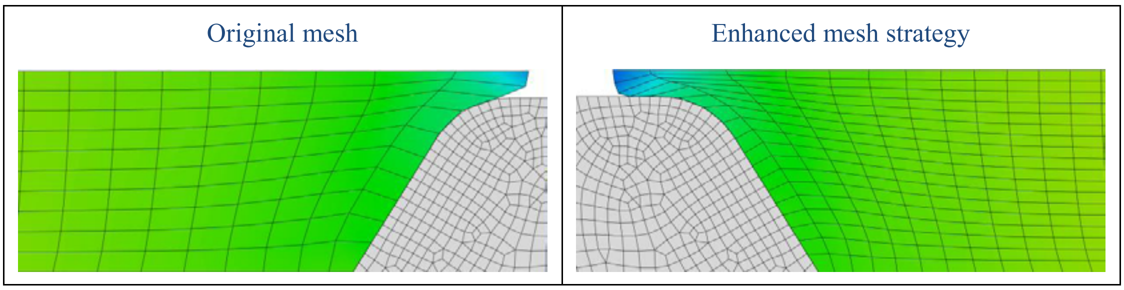

Figure 4: Mesh control during severe plastic deformation

3 SIMULATION RESULTS

3.1 Parametric mold design optimization

The shape of the mold and material sample, to a large extent determine the homogeneity of the stresses and the distribution of the strains in the sample. The depth of the cavities in the symmetric molds in a semi-constrained configuration has to be large enough to guarantee radial containment of the sample, on the other hand a too large depth can result in an undesirable contact of the molds at the end of the compression stage and/or during torsion. The depth of the sample cavity in the molds is chosen to be 0.3mm, which allows for a maximum sample compression of 0.4mm (1-2x0.3).

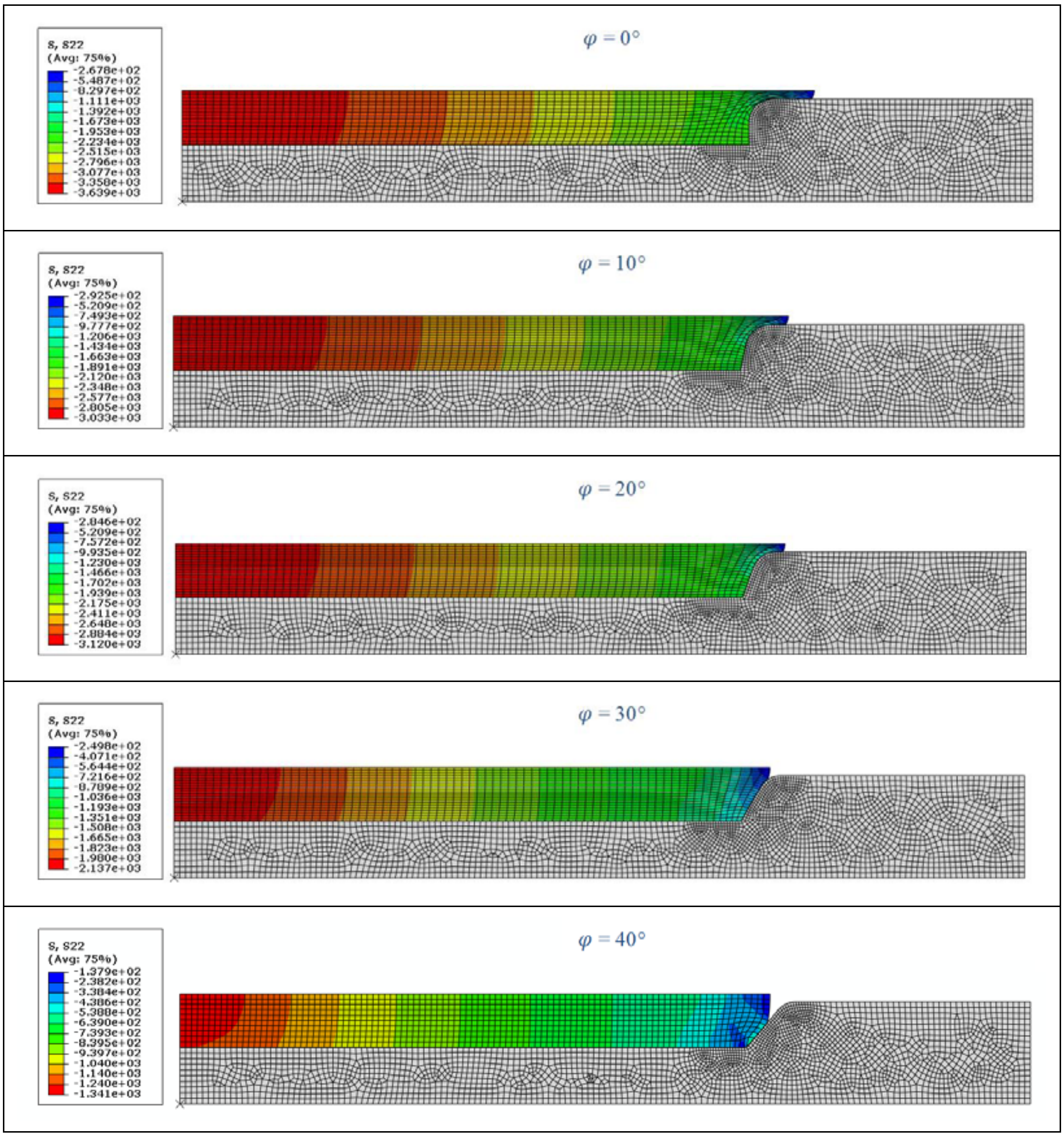

Figure 5: Influence of inclination on deformation and compressive stress state

Next to the depth of the mold cavity, also the inclination of the lateral boundary is important, since it affects both the level of hydrostatic pressure and strain distribution. The Abaqus/CAE integrated interface to enable the parametric design of the tool geometry is used to study the effect of specimen and mold dimensions. On Figure 5, the influence of the inclination on the sample deformation and the induced compressive stress state is clearly visible. The smaller the inclination, the higher the hydrostatic stress reached and the higher the homogeneity of the imposed stress.

3.2 High pressure torsion process

In Figure 6, the results of a HPT simulation using CGAX4 elements are shown, where the soft steel specimen has been subjected to a compressive load (corresponding to an 0.04 mm vertical die displacement), followed by a 30° rotation of the die.

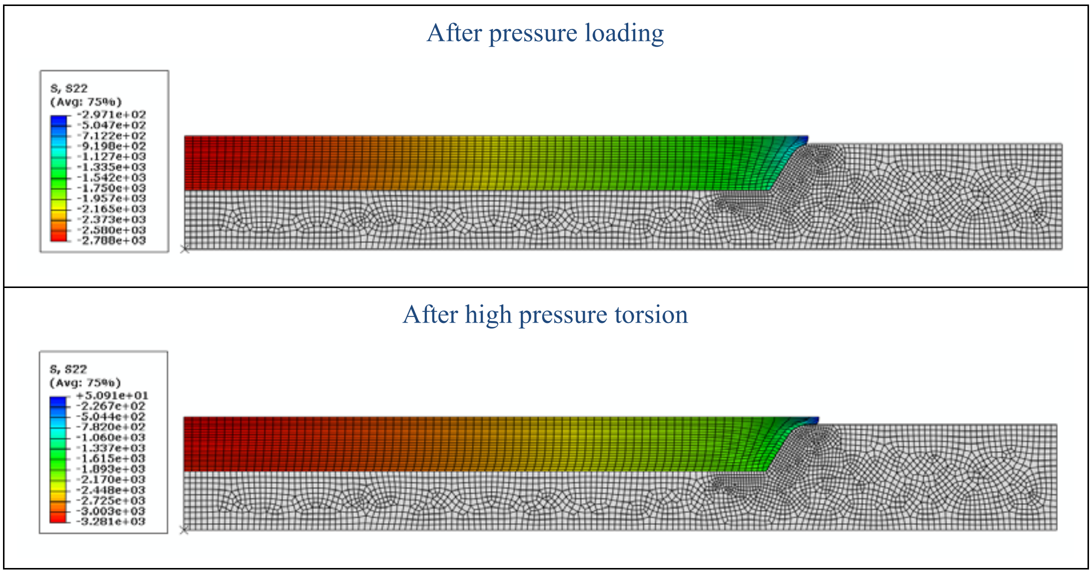

Figure 6: Sample deformation during high pressure torsion processing

On Figure 7, the predicted sample shape after high pressure torsion is shown. From this figure it is clear that during torsion the material flux outside the mold, which starts during the pressure stage, continues.

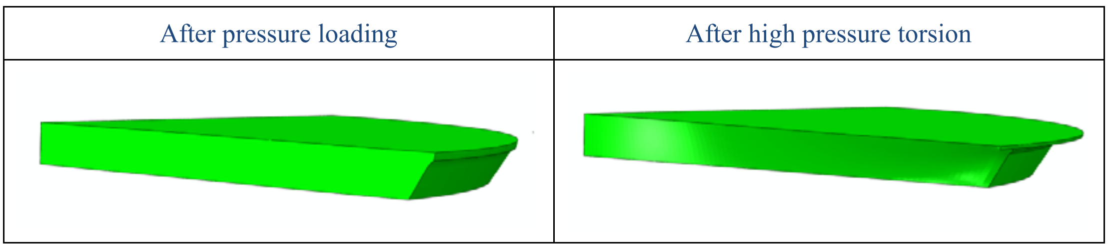

Figure 7: Prediction of the deformed sample shape

4 CONCLUSIONS

In this contribution simulations of the high pressure torsion process using the finite element code Abaqus are presented. Axisymmetric elements allowing twist (CGAX4) are computationally efficient to simulate the HPT process. Main challenges for the simulations are related to the extremely high levels of plastic strain to which the material sample is subjected. Different approaches are adopted. During the pressures stage a biased mesh combined with an adaptive remeshing technique is sufficient to cope with element distortion. However, for the torsion stage satisfactory results are only obtained by consecutive restart/remeshing steps. The obtained results allow us to come to an in-depth understanding of the sample response during severe plastic deformation and the parameters affecting HPT deformation.

REFERENCES

[1] Zhilyaev A.P. and Langdon T.G., Using High Pressure Torsion for Metal Processing: Fundamentals and Applications. *Progress in Materials Science* (2008) **53**:893-979.
[2] Draï A. and Aour B., Analysis of Plastic Deformation Behavior of HDPE during High Pressure Torsion Process. *Engineering Structures* (2013) **46**:87-93.
[3] Yoon S.C., Horita Z. and Kim H.S., Finite Element Analysis of Plastic Deformation Behaviour during High Pressure Torsion Processing. *Journal of Materials Processing Technology* (2008) **201**:32-36.
[4] Kim H.S., Finite Element Analysis of High Pressure Torsion Processing. *Journal of Materials Processing Technology* (2001) **113**:617-632.
[5] Kim H.S., Hong S.I., Lee Y.S., Dubravina A.A. and Alexandrov I.V., Deformation Behavior of Copper during High Pressure Torsion Processing. *Journal of Materials Processing Technology* (2003) **142**:334-337.