Journal of Physics: Conference Series

PAPER • **OPEN ACCESS**

Effect of punch force on the upsetting deformation process using three-dimensional finite element analysis

To cite this article: F.M. Mwema *et al* 2019 *J. Phys.: Conf. Ser.* **1378** 032094

Effect of punch force on the upsetting deformation process using three-dimensional finite element analysis

F.M. Mwema [1,2*], J.O. Obiko [3], E.T Akinlabi [1,4], S.A. Akinlabi [5] and O.S. Fatoba [1]

[1] Mechanical Engineering Science Department, University of Johannesburg, South Africa.
[2] Mechanical Engineering Department, Dedan Kimathi University, Kenya.
[3] Mining, Materials and Petroleum Engineering Department, Jomo Kenyatta University of Agric. & Techn., Kenya.
[4] Department of Mechanical Engineering, Covenant University, Ota, Ogun State, Nigeria
[5] Department of Mechanical and Industrial Engineering, University of Johannesburg, South Africa.
Corresponding Author; fredrick.mwema@dkut.ac.ke

Abstract-
Forging is one of the conventional shaping technologies that is widely used for the manufacture of quality products for various industrial applications. The process involves the mechanical application of a punching force to deform a material to the desired shape and improved properties. In most cases, the manufacture of quality products depends on the experience of the designer and trial and error method thus making the process wasteful and costly. The present study reports on the application of finite element method (FEM) for the analysis of the effect of punch force on the stress/strain distribution during the deformation process as a step towards the reduction of trial and error methods in practice. The results show that increase in the punch force leads to inhomogeneity in the strain/stress distribution due to change in the deformation temperature resulting from the internal heat generated during plastic deformation stage and the frictional force at the punch-workpiece interface. It is also observed that the maximum effective strain occurs at the center of the deformed sample and the maximum effective stress occurs at the low effective strain regions. Moreover, the friction parameter increases as the punch force increases.

Key words: Deform® 3D, finite element analysis, strain, stress, punch force.

1. Introduction

Metalworking processes are extensively used in industries to shape and improve the mechanical properties of functional components and materials [1,2]. The most commonly used processes for the manufacture of industrial parts include rolling, forging, extrusion, deep drawing and among others. The plastic deformation which occurs during metalworking processes can be classified as either bulk or sheet metal working. Bulk deformation involves deformation of the whole volume while sheet metal working involves localized deformation [3]. Bulk deformation processes such as forging (upsetting) involve large deformation and hence the analysis of deforming forces and stress can be complex [4].

For instance, the flow stress distribution during deformation largely depends on the deformation temperature, strain rate and strain in a forging process. In a laboratory setup, short cylinder compressed between two platens has been used to approximate the flow stress behavior of metals and alloys [5]. In recent times, Gleeble® thermal-mechanical simulator has been widely applied to simulate the industrial metalworking processes [6–9]. However, this process

encounters challenges in determining the optimum operating deformations due to the complexity involved during the deformation process. For example, the evolution of frictional forces be-tween the anvil and workpiece lead to the generation of internal heat hence affect the flow behavior [10]. Moreover, the frictional forces affect the flow pattern in which the middle section of the specimen flows outwardly leading to the phenome-non called barreling. This makes the production process complex and inhomogeneity during the deformation process. Most industrial production process largely depends on the experience of the designer and trial and error method, hence making them costly [1]. The development of finite element (FEM) computer software programs such as Deform® 3D, ANSYS, ABAQUS, etc. software has enhanced detailed evaluation of the influence of various process parameters on deformation, hence optimizing the forging process. In such cases, FEM simulation has been shown to reduce the cost and time of production [11]. Researchers and industries have extensively used FEM simulation to study various aspects of metal forming process, for the purpose of predicting metal flow behavior in the forging process [12–14]. Of the existing publications, Deform® 3D software has not been exhaustively used to study industrial processes such as forging except that it is widely used in severe plastic deformation studies [15-16]. Thus, in the present study, we illustrate the use of De-form® 3D simulation to predict the effect of punch force on the stress/strain distribution during a forging process.

.

2. Methodology

2.1 Theory of FEM Simulation

The finite element method (FEM) has been used to analyze the forging process based on the rigid-viscoplasticity using Variational principle. In this method, the stress and strain fields are calculated by the minimization of the functional variation shown in Equation 1.

$$\delta\varphi = \int_v \sigma_i \delta\dot{\varepsilon}_i dv + \int_v K\dot{\varepsilon}_v \delta\dot{\varepsilon}_v dv - \int_{SF} F_i \delta u_i ds = 0 \qquad (1)$$

In Equation 1, σ_i and ε_i are the effective stress and strain respectively, u_i the surface velocity components, SF is the surface force Fi the traction stress $\dot{\varepsilon}_v$ is the volumetric strain rate and K is a large positive penalty constant.

During an isothermal deformation process, the temperature distribution is assumed to be constant throughout the deformation process. However, in industrial practice, the isothermal process is unattainable due to the heat generated in the plastic deformation stage. Therefore, the temperature variation during the forging process can be calculated through the FEM using the Fourier formulation in Equation 2.

$$\nabla^T(K\nabla T) + \dot{q} = \rho C_p \frac{\delta T}{\delta t} \qquad (2)$$

In which K is the conductivity, T is the temperature, $\dot{q}$ the rate of heat generation during the plastic deformation stage, ρ is the density of the material under consideration, C_p the specific heat capacity and t is time [Eqn.2]. In a FEM simulation process, the effect of friction between the anvil-workpiece interface is mainly taken to be of a shear type (μ = 0.3). However, the

analysis has shown that friction is a function of strain and temperature. Thus, the frictional shear stress τ_f is expressed as shown in Equation 3.

$$\tau_f = -mk\left[\frac{2}{\pi}tan^{-1}\left(\frac{\vec{v}_s}{\mu_0}\right)\right]t \quad (3)$$

In Equation 3, m is the frictional coefficient factor that can be obtained from the specimen geometry ($0 \leq m \leq 1$), $\vec{v}_s$ is the surface velocity factor of the specimen to the anvil, k is the shear local flow stress, μ_0 is a constant ($\mu_0 << \vec{v}_s$) and t is the unit vector in the direction of the surface velocity vector.

2.2 Finite Element Analysis of Upsetting Process

Finite element analysis (FEA) provides a quick and accurate method to solve real-life problems. In the industrial application, FEA is used especially in shaping technology to approximate the expected metal flow behavior. It provides information on the response of the mechanical behavior of metals and alloys under an applied load. Therefore, the use of FEA eliminates the trial and error method in the production process and leads to a reduction in production cost. In this study, FEA analysis was used to evaluate the deformation characteristics of materials during upsetting process. The finite element model of short cylinder and platens were designed (in De-form® 3D tool) to have the same dimensions as the experimental specimens used in the Gleeble® 3500 thermo-mechanical simulators as reported in the literature [5]. The feed speed of the top die was 5 mm/s, which is mostly used in practice. The meshed and load-ed specimens are shown in Figure 1. The platens were set to be rigid bodies. The FEM simulation conditions and the physical properties of the deformed cylinder were: (a) the specimen (cylinder) was discretized into 202887 tetrahedron elements with a refined mesh of the whole volume (inside and outside surfaces) of the specimen, (b) to avoid friction, a lubricant is applied between the platen and the specimen during industrial upsetting; in the current simulation process the friction between the platen and the specimen interface was assumed to be of shear type and the friction factor was taken as 0.3, (c) the elastic deformation was neglected since during upset-ting high temperature and large deformations are involved (d) the surrounding temperature was assumed to be same as the room temperature of 25°C and the simultionn mode was isothermal, (e) the conventional coefficient to the environment was taken as 0.02 N/(s. mm. °C) and (f) the heat transfer coefficient between the platen and the deformed cylinder was taken as 5 N/(s. mm. °C).

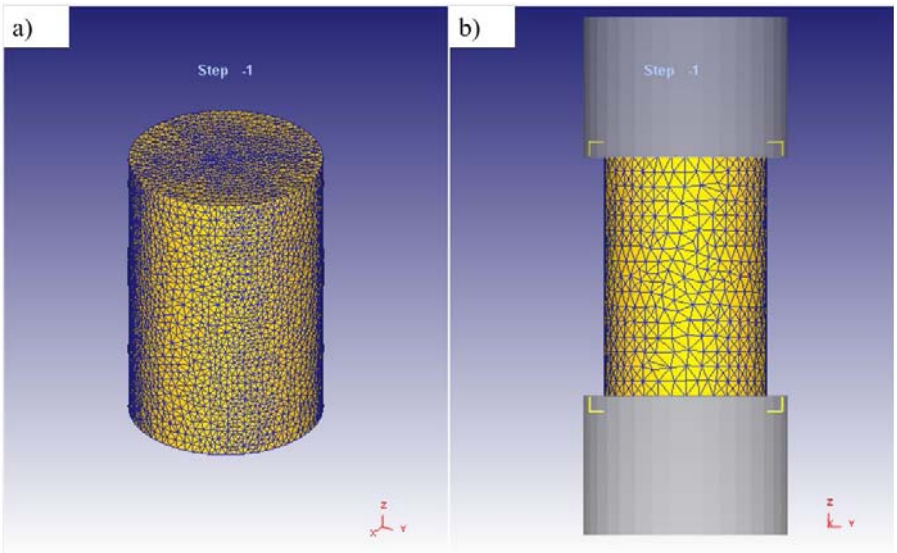

Figure 1: The finite element model (a) meshed specimen (b) meshed specimen under loading as prepared in Deform® 3D.

3. Result and discussions

3.1 The Effect of Punch Force on the Stress/Strain Distribution

The effect of punch force on strain/stress distribution in the deformed cylinder is as shown in Figure 2 after 100 simulation steps at 0.1s-1 strain rate and 800°C deformation temperature. It is shown that the strain distribution is non-uniform in the deformed cylinder with the maximum effective strain occurring at the center of the sample (Figure 2 (a)). This suggests that there is variation in the deformation behavior which might be attributed to changes in the deformation temperature during FEM simulation process (Figure 2(d)). Equbal et al. [17] investigated the deformation of micro-alloyed steel and reported non-homogeneous strain distribution with the maximum deformation occurring at the center of the specimen. Similarly, the effective stress distribution in the deformed cylinder exhibited non-homogeneous behavior and the maximum effective stresses were observed at areas of low strains (Figure 2(b)). In Figure 2(c), it is clearly seen that the positive maximum principal stress (tensile) was experienced at the circumference of the cylinder. In practice, these stresses can lead to deformation instabilities such as cracking and processing defects [5]. On the other hand, the negative maximum principal stresses (compression) occurred at the die-cylinder interface (Figure 2(c)). From the FEM simulation, it is evident that the stress/strain distribution inhomogeneities are closely associated with the change in deformation temperature resulting from internal heat generated during plastic deformation. As shown in Figure 2(d) at step 90, regions with high temperature experienced maximum effective strains and minimum effective stresses.

Figure 3 shows the stress/strain distribution of the specimen at a strain rate of 0.1 s-1 (at step 100). As shown, the deformation of the cylinder is heterogeneous, and the effective strain is located at the center of the deformed cylinder as observed at the strain rate of 0.1 s-1. Similarly, higher effective stress values were observed in the area with low effective strains. The standard deviation for effective strain and stress distribution as shown in Figures 3(a) and (b), further

affirm the heterogeneity of the strain and stress distribution during the forging of the cylindrical specimen.

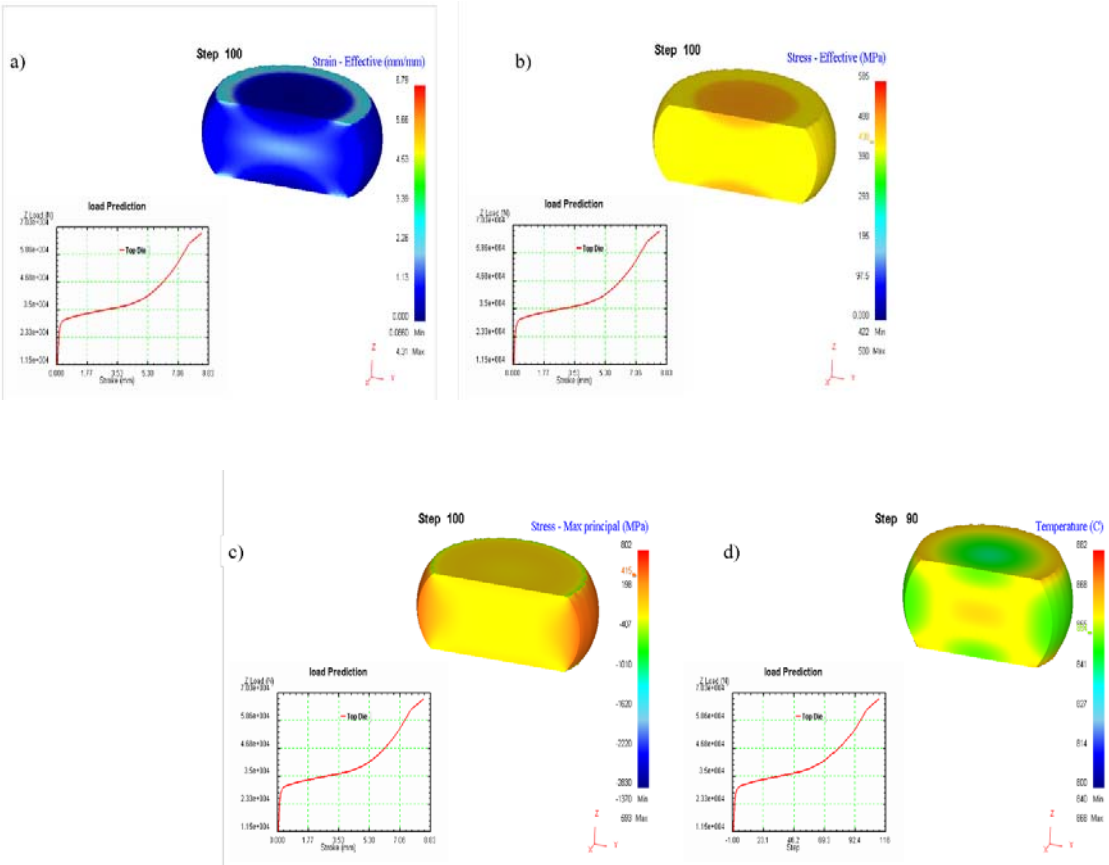

Figure 1: The effect of punch force on the (a) strain (b)stress (c) maximum principal stress (d) temperature distribution in the deformed cylinder at strain rate ($\dot{\varepsilon}$) 0.1 s-1. The load prediction curve and contour scales are also shown for each case.

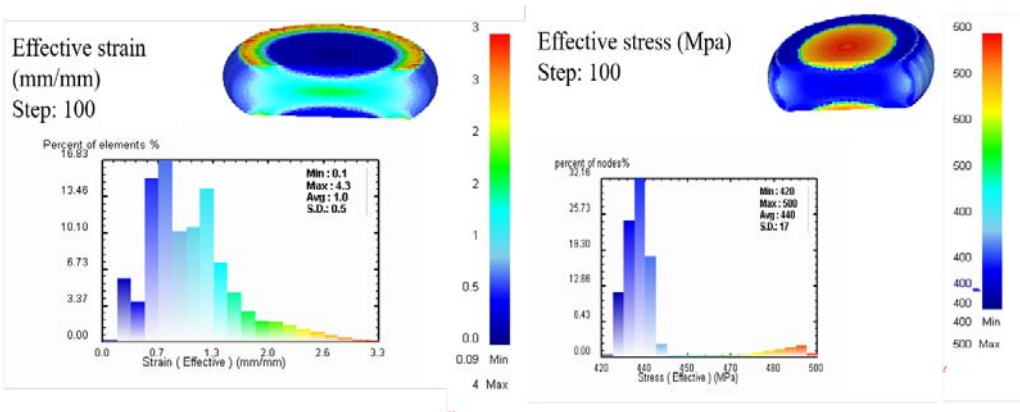

Figure 2: The stress/strain distribution of the deformed cylinder at strain rate simulation of 0.1 s-1 at step 100.

3.2 The Effect of Punch Force on Friction

Hot upsetting is a complex process and the quality of the product depends on the process parameters. The applied punch force leads to interfacial frictional forces between the die and the sample contact surfaces. As the friction force increases, the deformation temperature increases due to the heat generated during the plastic de-formation work. As such, the strain becomes inhomogeneous leading to temperature change and then the softening of the deforming material. This change affects the actual measured flow stress. Evans & Scharning [18] pointed out that the measured flow stress is not valid due to the changes in the deformation temperature as a result of the frictional heat generated during the deformation process, which leads to the increase in the internal temperature of the specimen. Consequently, the isothermal deformation process cannot be achieved due to the frictional heat generated between the anvil and the specimen [19]. Since the material flow properties are temperature and strain rate sensitive [18], the analytical solution to correct the flow stress with the effect of friction has been suggested (Equation (4)). Several researchers have adopted this equation to predict the actual flow stress during deformation process [19–21].

$$\bar{\sigma} = \frac{\sigma^a}{\left[1+\frac{2}{3\sqrt{3}} m \frac{r_0}{h_0} exp\left(\frac{3\bar{\varepsilon}}{2}\right)\right]} \tag{4}$$

Where, $\bar{\sigma}$ is the corrected flow stress, $\sigma^a$is the measured flow stress, $\bar{\varepsilon}$ is the measured instantaneous strain, h_0 and r_0 are initial height and radius of the specimen respectively and m is the friction factor determined according to the amount of barrelling coefficient determined using Equation (5). Further details on the determination of friction parameter, are discussed elsewhere [22].

$$m = \frac{3\sqrt{3}}{\left(3\frac{\Delta h}{\Delta R}\right) - 2\frac{h}{R}} \tag{5}$$

Where, ΔR is the difference between the maximum and minimum radius of the cylinder, Δh is the difference in height of the undeformed and deformed height of the specimen.

The shapes of the deformed materials simulated at 0.1 s-1 and 800°C at 70-100 deformation steps are as shown in Figure 4. For instance, it can be seen that at step 70 (Figure 3(a)) during the simulation the die was already in contact with the lateral side of the sample due to the increase in the friction coefficient. It was also observed that as the punch force increases there was an increase in the contact area at the die-sample interface (Figure 3). The increase in contact area was attributed to the gradual increase in the friction coefficient as the punch force increases [23-24]. The increase in friction parameter was quantified using Equation (2) as given in Figure 4. The sudden change in punch force observed in the load (N) against stroke (mm) slightly before or after step 70 can be attributed to an increase in the friction coefficient. It is of interest to note that, the instantaneous values of friction parameter obtained show that during the deformation process the coefficient of friction is no longer constant. The increase in friction coefficient increases the force required for deformation. Therefore, during tool and die designing process it is important to consider the effect of friction to predict the actual required deformation force.

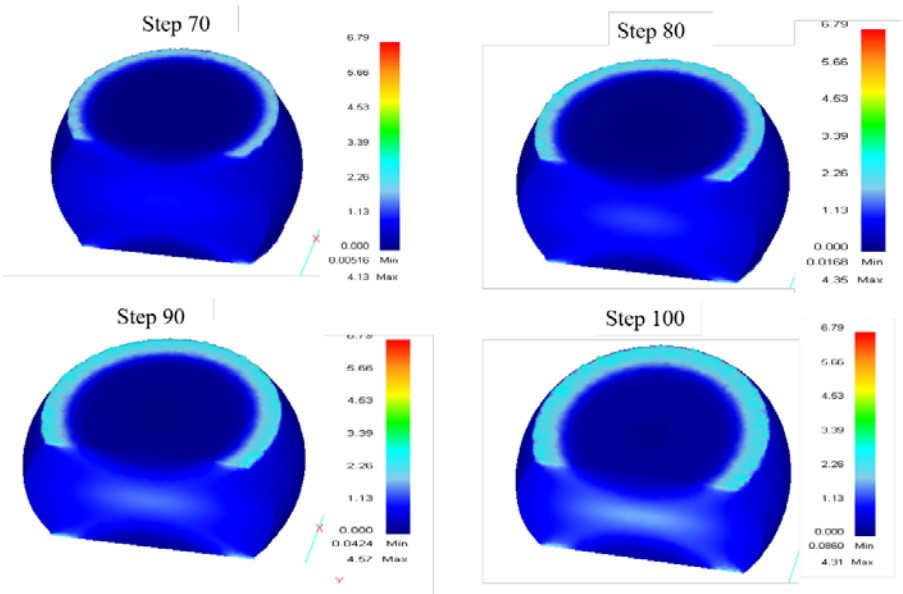

Figure 3: The effective strains (mm/mm) of the sample at deformation steps of (a) 70 (b) 80 (c) 90 and (d) 100 FEM simulation at 800°C and 0.1 s^{-1}.

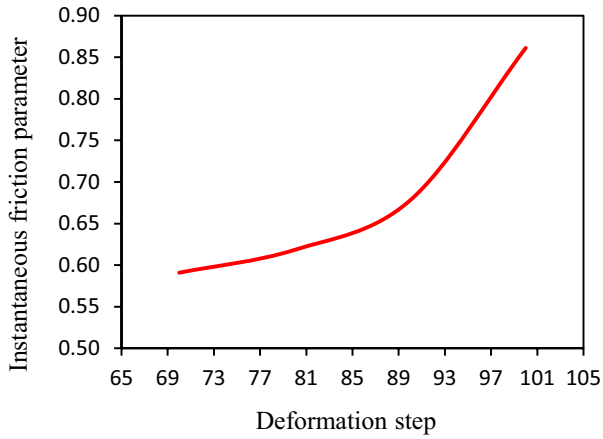

Figure 4: The variation of friction parameter at different deformation step for FEM simulation at 800°C and 0.1s^{-1}.

4. Conclusion

The FEM simulation analysis of the forging process was conducted using Deform 3D software. The study has shown that punch force significantly influences the stress/strain distribution. The variation of the effective stress/strain across the cross-section of the deformed cylinder was attributed to an increase in the friction coefficient at the punch/workpiece interface. Therefore, it was concluded that the coefficient of friction was not constant for the entire compression process as assumed. The frictional force in the punch/workpiece interface influences the deformation behavior and changes the profile of the free surface of the deformed cylinder. Numerical calculations show that the friction coefficient resulting from the punch/workpiece inter-face increases exponentially as the strain increases. Thus, the friction effect leads to inhomogeneous effective stress/strain distribution in the deformed workpiece. From these

results, it can be concluded that inhomogeneous mechanical properties are achieved during the forging process. Therefore, Deform® 3D simulation is an effective and efficient method to approximate the expected mechanical properties and thereby optimizing the forging process. The method reduces the number of experiments and manufacturing costs.

Reference

[1] Maarefdoust, M. (2012). Simulation of finite volume of hot forging process of industrial gear. In: International Conference on Networks and Information, pp. 111–115. IACSIT Press, Singapore.
[2] Vostrov, V.N., Kononov, P.V. (2016). Finite-Element Simulation of Flanging in the Deform 3D Software Package. *Russian Metallurgy (Metally)*, 2016(5), 461–466.
[3] Beddoes. J., Bibby, M.J. (1999). Principles of Metal Manufacturing Processes. Elsevier, USA.
[4] Dieter, G. (1986). Mechanical Metallurgy. 3rd Edn. McGraw Hill, New York.
[5] Yang, Z.N., Dai, L.Q., Chu, C.H., Zhang, F.C., Wang, L.W., Xiao, A.P. (2019). Effect of Aluminum Alloying on the Hot Deformation Behaviour of Nano-bainite Bearing Steel. *JMEPEG*, 26, 5954-5962.
[6] Mehtonen, S.V., Karjalainen, L.P., Porter, D.A. (2013). Materials Science & Engineering A Hot deformation behavior and microstructure evolution of a stabilized high-Cr ferritic stainless steel. *Mater. Sci. Eng. A*, 571, 1–12.
[7] Pu, E., Zheng, W., Xiang, J., Song, Z., Li, J. (2014). Hot deformation characteristic and processing map of superaustenitic stainless steel S32654. *Mater. Sci. Eng. A*, 598, 174–182.
[8] Liu, X.G., Ji, H.P., Guo, H., Jin, M., Guo, B.F., Gao, L. (2013). Study on hot deformation behaviour of 316LN austenitic stainless steel based on hot processing map. *Mater. Sci. Technol.*, 29(1), 24–29.
[9] Shu-li, H., Min-gang, S., Wen-wu, Z. (2010). Hot Deformation Behavior and Hot Processing Map of P92 Steel Shu-li SUN. *Mater. Adv. Publ. Trans Tech*, 101, 290–295.
[10] Lin, Y.C., Chen, M.S., Zhong, J. (2009). Effects of deformation temperatures on stress/strain distribution and microstructural evolution of deformed 42CrMo steel. *Material & Design*, 30(3), 908–913.
[11] Yang, C., Lin, X. (2016). The Forming Analysis of Two-stage Extrusion for 1010 Fastener. *Journal of Mechanical Engineering and Automation*, 6(3), 43–50.
[12] Lee, Y.S., Lee, S.U., Van Tyne, C.J., Joo, B.D., Moon, Y.M. (2011). Internal void closure during the forging of large cast ingots using a simulation approach. *J. Mater. Process. Technol.*, 211(6), 1136–1145.
[13] Zhang, Z.J., Dai, G.Z., Wu, S.N., Dong, L.X., Liu, L.L. (2009). Simulation of 42CrMo steel billet upsetting and its defects analyses during forming process based on the software DEFORM-3D. *Mater. Sci. Eng. A*, 499(1–2), 49–52.
[14] Na, Y.S., Yeom, J.T., Park, N.K., Lee, J.Y. (2003). Simulation of microstructures for Alloy 718 blade forging using 3D FEM simulator. *J. Mater. Process. Technol.*, 141(3), 337–342.
[15] Mwema, F.M., Kiptoo, M., Shagwira, H., Obiko, J. (2017). Design Optimization of Angular Extrusion for Severe Plastic Deformation of Tubular Specimens. *International*

Journal of Mechanics and Applications, 7(1), 1-13.
[16] Zhai, D., Osman, M., Li, L., Zheng, Y., Tong, Y. (2016). Simulation and Experimental Investigation for the Homogeneity of Ti49.2Ni50.8 Alloy Processed by Equal Channel Angular Pressing. *Metals*, 6(45).
[17] Equbal, M.I., Talukdar, P., Kumar, V., Ohdar, R.K. (2014). Deformation Behavior of Micro-alloyed Steel by Using Thermo Mechanical Simulator and Finite Element Method. *Procedia Mater. Sci.*, 6, 674–681.
[18] Evans, R.W., Scharning, P.J. (2001). Axisymmetric compression test and hot working properties of alloys. *Mater. Sci. Technol.,* 17(8), 995–1004.
[19] Qian, L.Y., Fang, G., Zeng, P., Wang, L.X. (2015). Correction of flow stress and determination of constitutive constants for hot working of API X100 pipeline steel. *Int. J. Press. Vessel. Pip*., 132–133, 43–51.
[20] Gholamzadeh, A., Taheri, K. (2009). The prediction of hot flow behavior of Al-6%Mg alloy. *Mech. Res. Commun*., 36(2), 252–259.
[21] Mostafaei, M.A., Kazeminezhad, M. (2012). Hot deformation behavior of hot extruded Al-6Mg alloy. *Mater. Sci. Eng. A*, 535, 216–221.
[22] Luan, J., Sun, C., Li, X., Zhang, Q. (2014). Constitutive model for AZ31 magnesium alloy based on isothermal compression test. *Mater. Sci. Technol.,* 30(2), 211–21.
[23] Hu, M., Dong, L., Zhang, Z., Lei, X., Yang, Y., Sha, Y. (2018). Correction of Flow Curves and Constitutive Modelling of a Ti-6Al-4V Alloy. *Metals*, 8(4), 256.
[24] Li, Y.P., Onodera, E., Matsumoto, H., Chiba, A. (2009). Correcting the stress-strain curve in hot compression process to high strain level. *Metall. Mater. Trans. A Phys. Metall. Mater. Sci.*, 40(4), 982–990.