# Experimental and Numerical Simulation for Bulk Nanostructure of AL-6082 Alloy Material Produced by Equal-Channel Angular Pressing (ECAP)

**Muamar M. Benisa**
*Mechanical Engineering Department*
*University of Asmarya, Libya*
muamarisa@yahoo.com

**Bashir S. Younise**
*Mechanical Engineering Department*
*University of El-Mergib, Libya*
bsyounise@elmergib.edu.ly

**Hitem A. Aswihli**
*Mechanical Engineering Department*
*University of Asmarya, Libya*
haswihli@asmarya.edu.ly

*Abstract*—**Equal-channel angular pressing (ECAP) is a severe plastic deformation (SPD) method, which used to produce ultra fine-grained (UFG) material . The conventional way to develop press-formed metallic components requires a burden some trial-and-error process for setting-up the technology, whose success depends largely on the operator's skill and experience [6,7]. The finite element (FE) simulations of ECAP process help the manufacturing engineer to design a forming process by shifting the costly press-shop try-outs to the computer-aided design environment. Numerical simulations of a manufacturing process, such as ECAP, have been introduced, in order to avoid the trial-and-error procedure and to shorten the development phases when tight time-to-market is demanded. The main aim of the investigation presented in this paper is to develop a numerical model that would be able to successfully simulate the ECAP pressing. The finite-element method was used for die and billet-behavior predictions during the process. The as-received alloy used in this study was the 6082 commercial Al-Mg-Si alloy. High strength and high ductility phenomenon which recently was found in materials after ECAP were reached with the route C technique up to eight passes with a high length to diameter ratio i.e. 15-16. The induced anisotropy of specimens after ECAP was monitored.**

*Index Terms*: **nanostructure materials, FE simulation, equal angler pressing (ECAP).**

## I. INTRODUCTION

ECAP is one of the most wide-spread techniques for producing ultra-fine grained materials. The main idea of ECAP is that the metal is deformed through a process of simple shear taking place in the cross sectional area without any geometrical change of the ingot [1-7,10,11]. Methods of severe plastic deformation ought to meet a number of requirements, which should be taken into account while developing them to form nanostructures in bulk samples and billets.

These requirements are as follow: Firstly, it is important to obtain ultra fine-grained structures with prevailing high-angle grain boundaries since only in this case a qualitative change in properties of materials could occur. Secondly, the formation of nanostructures uniform within the whole volume of a sample is necessary for providing stable properties of the processed materials. Thirdly, though samples are exposed to large plastic deformations they should not have any mechanical damage or cracks [1, 2, 3, 5]. The basic principles of ECAP have been delineated in several detailed reports [1-6]. Briefly, the die is constructed containing two channels, equal in cross-section, intersecting at an angle which is generally close to 90°. The work-piece is machined to fit within the channel and it is pressed through the die (the material is subjected to deformation which approximates to simple shear) [1,2]. Several studies dealt with different aspects of this process from structural change to finite element (FE) process modeling [2-5]. In this paper, FE analysis was used to optimize the technology of ECAP. On the basis of FE modeling results, long homogeneous billets could be manufactured by the ECAP process. Therefore, in this work, numerical simulations were used to optimize the technology of ECAP process. To find optimal model of Non-linear FE analysis was conducted to predict stress and strain distributions, and forming forces during the ECAP process. The main goal was to develop a computer model that would be able to simulate ECAP, and therefore, to achieve the reliable and accurate enough model, so that even the try-out tools and the time-consuming try-out processes may be eliminated, or at least significantly reduced.

## II. FINITE-ELEMENT MODELING AND EXPERIMENTAL VERIFICATION

In this work, the commercial finite element software ANSYS was used to approach FE simulation. In order to reduce the processing time and improve the precision of calculations, 3D ax-symmetric FE models were created for a bulk nanostructure material with a high length to diameter ratio as 15-16 and analysis were carried out.

---

Received 28 Oct, 2019; revised 24 Nov, 2019; accepted 4 Dec, 2019.

The models in FE analysis included two elements only: a rigid die and a billet of material. The die was modeled as a rigid body, because of the fact that the die material (Steel) is much less deformable than the billet material (Aluminum). So, the stress and the strain of the die were not analyzed. As a result, neither the material properties concerning to the die were important, nor the mesh was generated. This procedure eliminated unnecessary calculations that might cause a decrease in both run time and errors in the numerical solution. The deformable material (AL-6082) in FE models have been modeled with Plane 183 finite element. Plane 183 has quadratic displacement and plasticity. Multilinear isotropic hardening material in ANSYS workbench was applied into aluminum billet to simulate large plastic strain deformation. Von Mises yield criterion coupled with isotropic work hardening assumption were applied too. The interface contacts of the billet- die were modeled as deformable, and the ANSYS software was used to solve these tasks on the basis of the contact-target-surface approach with an adjustable impenetrability constraint that assures contact compatibility. CONTA171 and TARGE169 have been created at interface contacts. CONTA171 is used to represent contact and sliding between 3-D "target" surfaces (TARGE169) and a deformable surface, defined by this element. The element is applicable to 3-D structural and coupled field contact analysis. TARGE169 is used to represent various 3-D "target" surfaces for the associated contact elements CONTA171. The contact has been presented on the circumference and the lower base of the billet. On the other hand, the target has been presented on the channel of the die. The number of nods and elements used was 323282 and 226211 respectively.

In the FE model, the die was modeled as a rigid body and the mesh was not generated as shown in Figure1. Remote displacement was applied on the outer circumference of the die, while displacement constraint was applied on the upper side of the aluminum billet (Figure2).

In this work, a cylindrical billet of an Aluminum alloy (AL-6082) with 15 mm diameter and a length of 230mm was used as a deformable martial. The material properties (stress - stain curve of billet material) were determined from the tensile test as shown in Figure3.

The elastic module of the billet material (E) is 71GPa and its Poisson’s ratio (v) is 0.334. The frictional behavior between the billet and the die channel was assumed to be following coulombs model. The friction coefficient was considered to be 0.05.

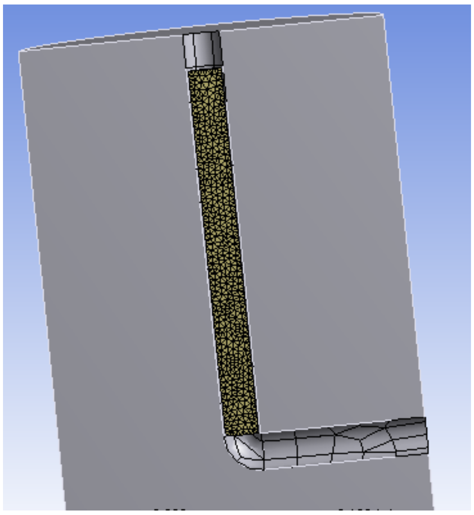

Figure 1. Rigid Die and Billet FE Mesh Symmetrical Models Used for ECAP.

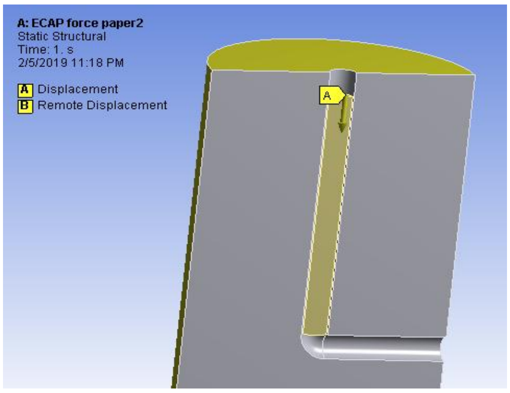

Figure 2. Constrain Used in the FE Simulation.

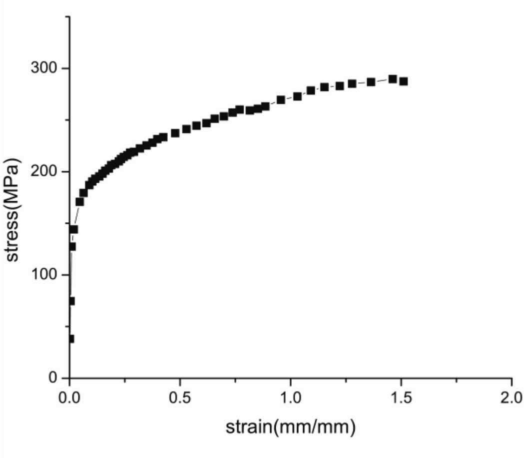

Figure 3. Experimental Tensile-Stress-Strain Curve for the Aluminum Billet.

In order to validate the FE simulations results, the Equal-Channel Angular pressing (ECAP) experiments were carried out. An experimental set-up (shown in Figure 4) was used. The die was made of steel and a hydraulic press machine with the maximum capacity of 160 ton was used in this work [1,2]. The well lubrication of the cylindrical billets of 15 mm in diameter and 230 mm in length were placed into the vertical channel, and then the punch extrudes a billet through an ECAP die set having 90º inter-sectioning channels with identical cross-section. One, four and eight passes were performed by route C (rotation of the billets around its longitudinal axes after each pass by 180º clockwise) as mentioned above. An experimental setup and schematic representation of ECAP die are illustrated in Figure 4.

The ECAP process is performed at room temperature with a constant displacement rate of 8 mm/min [1,2 , 8]. The length to diameter ratio of specimens is relatively high (more than 15). Under these conditions the sample moves inside the channels (a rigid body) and deformation is achieved only at the intersection channel by simple shear [2-5,7,8,9]. After the extrusion stroke, the punch returns to its initial position.

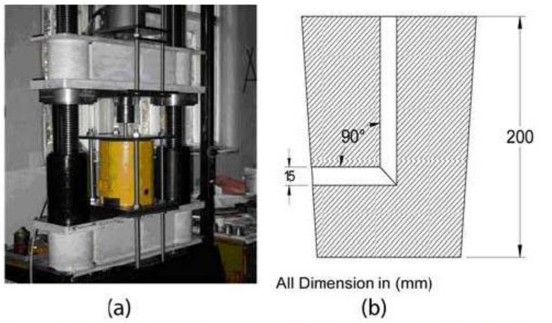

Figure 4. (a) Experimental Tool Set Up ,(b) Schematic of ECAP Die.[2]

## III. RESULTS AND DISCUSION

As mentioned above, the forming force was presented as a displacement applied on the upper side of the aluminum billet (Figure2). The convergences of the forming forces for ECAP model, obtained through the FE simulations, are shown in Figure5. This figure shows that the highest value of a forming force is (37833N). It can be seen that the magnitude of the forming force increases as the simulation time increases (as the punch extruded the billet through the inter-sectioning channel).

The FE simulation of the ECAP process passes through three stages/steps (corresponding to the forming process). During the first step – the punch pushes the billet to the end of the vertical channel and starts to slide into the second channel (almost at inter-sectioning channel), so the forming load is very small (Figure 5).The time needed for this step is short (between 0.07 sec and 0.14 sec from simulation time – Figure 5 ). After 0.14 sec, the second step starts and the forming load decreases. During this step, the front of the billet enters into the second channel (horizontal channel). The time needed for this step was short (between 0.14sec and 0.21sec from the simulation time – Figure5). After 0.21 sec, the third step starts and the forming load increases sharply, at this time the punch extrudes the billet to the second channel, in where the fine - grain structure takes place in the whole billet (Figure5).

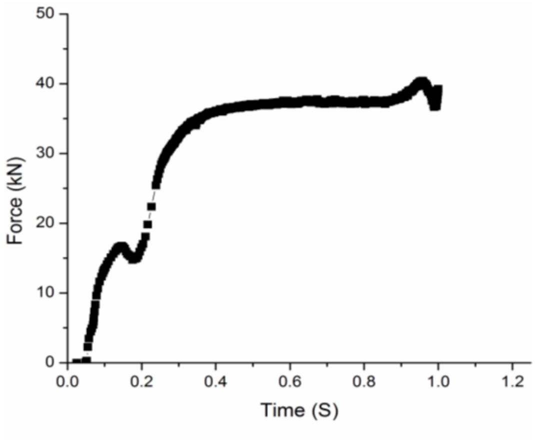

Figure 5. Forming Force for FE Model of ECAP.

Moreover, at this step (third step) the billet has a maximum equivalent stress (approximately between 0.27sec and 0.42sec from the simulation time), after that the equivalent stress has a slight decrease. After 0.42 sec, the stress value has a constant level as shown in Figure 6 (b). On the other hand, the maximum plastic strain could be pointed at 0.27sec from the simulation time, as shown in the Figure 7(a). After about 0.3 sec of the simulation time, the plastic strain decreases sharply. At this time, the front portion of the billet takes place in the horizontal channel, which leads to an easy sliding of the billet in the horizontal channel as shown in Figure 6 (a) and Figure 7(b).

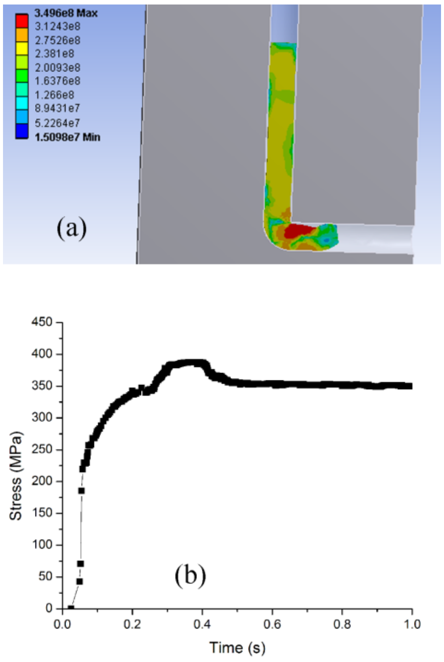

Figure 6. (a) FEM Modeling Simulation,(b) Equivalent Stress During ECAP.

Figure (8) illustrates the 3D symmetry of the total deformation of the ECAP FEM model, and it also shows the final ECAP billet in the final stage (step) of the ECAP process.

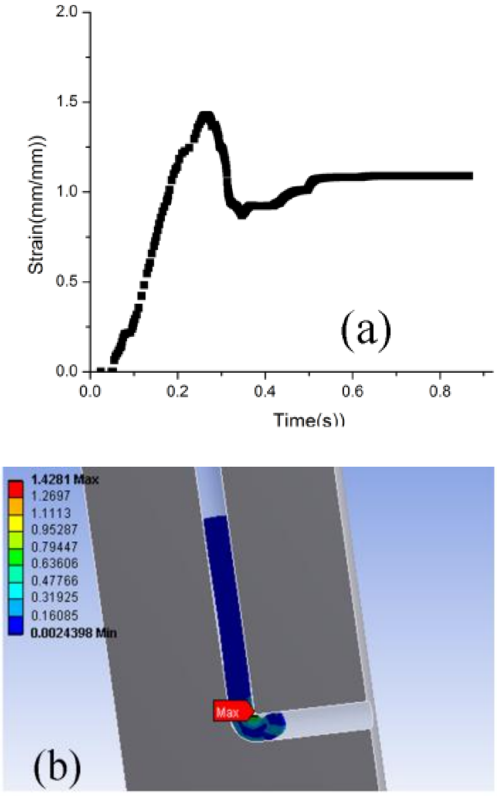

Figure 7. (a) Equivalent Plastic During ECAP Simulation, (b) FEM Modeling

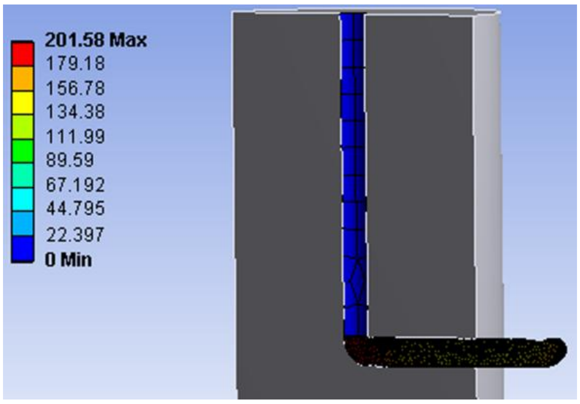

Figure 8. Total deformation of ECAP simulation model.

Figure 9(a) shows the 3D symmetry model of the ECAP formed billet as obtained from the simulation. This was verified by the experiment as it can be seen in the Figure 9(b), which shows the billets after different passes, from left, zero, one, four, and eight passes. In Figure (9), it can be seen that the ECAP numerical model of the billet is very close to the experimental one, especially after one pass. Also, the experimental and numerical results were in a quite good matching as well.

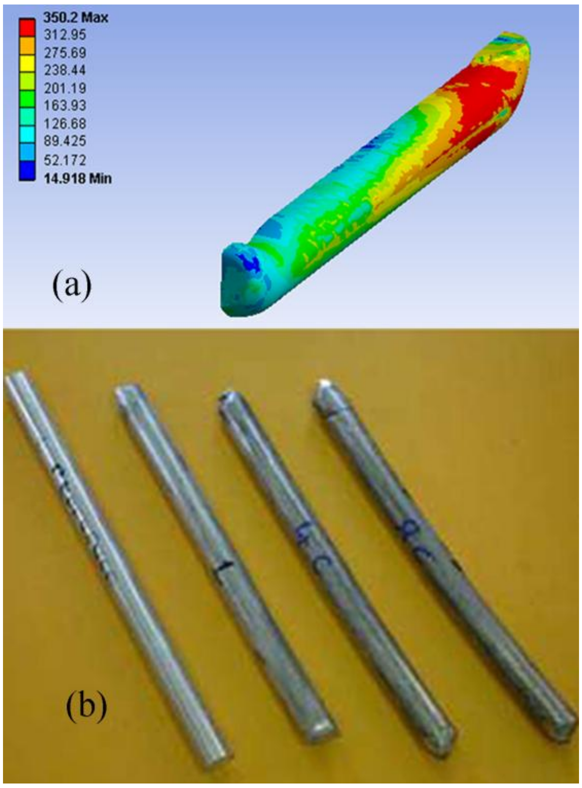

Figure 9. Billets after ECAP (a) FEM Simulation and (b) Experiment , From Left: Zero Pass, One Pass, Four Passes, Eight Passes, Respectively.

## IV. CONCLUSION

A finite-element simulation of the ECAP could be a very useful tool for understanding and improving forming operations, because it provides important data to determine the forming parameters and the operation time. The developed FE models and the method proposed in this paper have been proved to be sufficiently effective in predicting the final shape of the ECAP component and the regions of any sustainable mechanical damages or cracks during the process. However, it would be noticed that the optimization procedure of the press-forming processes (ECAP) – owing to the presence of the hardly reproducible phenomena like friction and lubrication – would not be limited to simple numerical simulations. Due to the fact that this phenomena can contribute a lot towards saving costs and reducing the time-to-market, it is currently held up by empirical trial-and-error processes. Forming-simulation results, today, are reliable and accurate enough so that even the tryout tools and the time-consuming try-out processes may be eliminated, or at least significantly reduced.

## REFERENCES

[1] M. M. Benisa, F. F. Eldabee and A. A. Elderwish "*The effects of backward extrusion on bulk nanostructured AL-6082 materials produced by equal-channel angular pressing (ECAP),* Journal of Sciences and Technologies (Engineering and Applied Science ), Vol. (3), No.(1),pp.52-65, June,2019.

[2] M. Benisa, G. Senuss, "*Investigation of mechanical properties for bulk nanostratural of AL-6082 alloy material produced by equal-channel angular pressing (ECAP),* " Structural integrity and life, Vol. 14, No(2), pp. 79.86, 2014.

[3] Gy. Krallics, D. Malgin, A. Foder, "*Experimental Investigations of the Al 6082 Alloy Subjected to Equal-Channel Angular Pressing,*" Materials Science Forum, vol.473 , pp.129-134, 2005

[4] Gy. Krallics, Z. Szeles, D. Malgyn, "*Finite Element Simulation of Multi-Pass Equal Channel Angular Pressing,"* Materials Science Forum, , Vols. 473-474, pp. 453-458, 2005.

[5] R.Z. Valiev, R.K. Islamgaliev, I.V. Alexandrov, "*Nanostructured Materials From Several plastic Deformation: Nanostructured Materials,* " Vol. 12, pp. 35-40, 1999.

[6] R.Z. Valiev, R.K. Islamgaliev, I.V. Alexandrov "*Bulk Nanostructured Materials From Severe Plastic Deformation,* " Progress in Material Science,Vol. 45, pp. 103-189, 2000.

[7] R.Z. Valiev and I.V. Alexandarov: "*Nanostructured Materials from sever plastic deformation*," Nanostructured Mater; pp. 12-35. 1999.

[8] R.Z. Valiev, Yu.V. Ivanisenko, , E.F. Rauch, B. Baudelet, "*structure and deformation behavior of Armco iron subjected to severe plastic*," Acta Mater. 44:4705. 1997.

[9] M. Furukawa, Z. Horita, M. Nemoto, T. Langdon, "*A model investigation of the shearing characteristics in equal-channel angular pressing,"* Journal of Material Scince 36, pp. 2835-2843, 2001.

[10] V. M .Segal, "*Matreials processing by simple shear*," Mater Sci. Eng; A197: 157, 1995.

[11] Y. Iwahashi, Z. Horita, M. Nemoto, T. G. Langdon, "*influence of channel angle on the development of ultrafine grains in equal-channel angular pressing*," Acta Mater., 46:1589, 1998.

## BIOGRAPHIES

**Muamar M. Benisa** was born in Zliten/ Libya on March 13,1970. He received BSc degree in Mechanical Engineering from University of Garyounis (Benghazi-Libya), in 1992, He got MSc degree in Production Engineering from Budapest university/Hungary in 2005.

Moreover, he got PhD degree in Production Engineering from Belgrade University /Serbia, in 2013, where currently he has worked in Department of Mechanical Engineering at *Asmarya* University /Libya.

**Bashir S. Younise** was born in Alkhoms/ Libya on July 06, 1969. He received BSc degree in Mechanical Engineering from University of Tripoli in 1993, He got MSc degree in Production and materials Engineering from Budapest university/ Hungary in 2007. Moreover, he got PhD degree in Materials Engineering from Belgrade University /Serbia, in 2013, where currently he has worked in Department of Mechanical and Industrial Engineering at El-Mergib University /Libya.

**Hitem A. Aswihli,** born on April 26, 1985 in Zliten, Libya. He has received a Higher Diploma in Mechanical Engineering from the Higher Institute of Marine Fisheries, Sabratha, in 2005. Also he had been awarded a Bachelors degree in Industrial Engineering from the Faculty of Industrial Technology, Misurata, in 2009. Moreover, he got an MSc degree in Manufacturing Engineering from the University of Manchester in 2014. Worked as a lecturer in the Faculty of Engineering at the *Asmarya* Islamic University, Libya since 2016 until the present date.