The International Journal of Engineering and Information Technology (IJEIT)
ijeit.misuratau.edu.ly
جامعة مصراتة MISURATA UNIVERSITY

A Numerical Simulation and Experimental Validation of Backward Extrusion for AL-6082 Alloy Material

Muamar M. Ben Isa[1] m.benisa@asmarya.edu.ly, Faraj F. Eldabee[2] f.edabee@uot.edu.ly, Nasserdeen Ashwear[3,*], nasseradeen-ashwear@eng.misuratau.edu.ly, Ahmad E. Alaian [4] a.aian@asmarya.edu.ly, Mahmud Eljaarani[5] m.eljaarani@mu.edu.ly

[1,4]Mechanical and Industrial Engineering Department, Al Asmarya Islamic University, Zliten, Libya
[2]Mechanical Engineering Department, University of Tripoli, Tripoli, Libya
[3]Mechanical Engineering Department, Misurata University, Misurata, Libya
[5]Industrial & Manufacturing Engineering Department, Misurata University, Misurata, Libya
* Corresponding author

Article History
Received 24 Sep, 2024
Revised 13 Nov, 2025
Accepted 15 Nov, 2025
Online 10 Dec, 2025

DOI:
https://doi.org/10.36602/ijeit.v14i1.584

Index Terms
Backward extrusion, Aluminum, Finite Element Simulation.

Abstract

Globally, the backward extrusion process plays a significant role in manufacturing industries. When designing forming processes, manufacturing engineers can use Finite Element (FE) simulations of backward extrusion, which replace expensive press-shop try-outs with a computer-aided design environment. This paper's primary goal is to create a numerical model that can accurately replicate the reverse extrusion process. Finite Element software ANSYS will be used to predict billet behavior during the backward extrusion process, helping to eliminate trial-and-error procedures and thus shorten development phases when tight time-to-market deadlines are required. The simulation and analysis of a number of important backward extrusion process parameters, including forming force, stress, and strain distribution in a billet, are the main objectives of this work. A commercial Al-Mg-Si alloy (6082) is used in this study. Experimental results demonstrate the effectiveness of FE simulations, with a strong focus on backward extrusion production. There is a good agreement between the forming load values obtained from the finite element simulation and the experimental procedure.

المحاكاة العددية والتحقق التجريبي لعملية البثق العكسي لسبيكة الألمنيوم Al-6082

معمر بن عيسى[1]، فرج الضبيع[2]، نصرالدين الشويعر[3،*]، أحمد العيان[4]، محمود الجعراني[5]

[4،1] قسم الهندسة الميكانيكية والصناعية، الجامعة الاسمرية الاسلامية، زليتن، ليبيا
[2] قسم الهندسة الميكانيكية، جامعة طرابلس، طرابلس، ليبيا
[3] قسم الهندسة الميكانيكية، جامعة مصراتة، مصراتة، ليبيا
[5] قسم الهندسة الصناعية والتصنيع، جامعة مصراتة، مصراتة، ليبيا
* المؤلف المراسل

الكلمات المفتاحية
البثق العكسي، الألمنيوم، محاكاة العناصر المحدودة.

الملخص

تلعب عملية البثق العكسي دوراً مهماً في الصناعات التحويلية وبشكل كبير. بإمكان مهندس التصنيع توظيف محاكاة العناصر المحدودة **(FE)** للبثق الخلفي في تصميم عمليات التشكيل عن طريق تحويل تجارب الضغط المكلفة إلى بيئة التصميم بمساعدة الكمبيوتر. إن الهدف الرئيسي من هذه الورقة هو تطوير نموذج رقمي من شأنه أن يحاكي بنجاح عملية البثق العكسي. تم استخدام برنامج العناصر المحدودة **ANSYS** للتنبؤ بسلوك الكتلة المعدنية أثناء عملية البثق العكسي، الأمر الذي قد يؤدي إلى تجنب إجراء التجارب وحدوث الأخطاء ومن ثم تقليص مراحل التطوير وكسب الوقت. تهتم هذه الدراسة بالمحاكاة والتحقق من عدة عوامل مهمة والمرتبطة بعملية البثق العكسي (مثل قوة التشكيل والإجهاد وكذلك توزيع الانفعال في الكتلة المعدنية). تم استخدام سبيكة (6082) **Al-Mg-Si** التجارية في هذه الدراسة. أظهرت النتائج المعملية فعالية المحاكاة **FE** في عملية البثق العكسي. حيث كان هناك توافق جيد في قيم حمل التشكيل لمحاكاة العناصر المحدودة مع القيم المتحصل عليها من التجارب العملية.

I. INTRODUCTION

Extrusion has gained a lot of attention lately in the production of numerous goods made of various materials, especially in the aluminum sector. The basic idea behind the extrusion process is rather straightforward: a workpiece (billet) is put in a closed container and then forced through a die using a device known as a ram [1]. A successful final processing method for creating fine-grained goods is the backward extrusion technique [2]. As a result, this method has emerged as one of the most promising manufacturing procedures for a variety of reasons, including low machining requirements, appropriate stress distributions, and material savings. At present, aluminum alloys could be considered as the most widely used structural materials in numerous industries such as aeronautics and aerospace industry [3]. Furthermore, when compared to other manufacturing techniques, the backward extrusion process offers a number of benefits. Low material usage, good dimensional precision with suitable finishing, adequate mechanical and microstructural qualities, and the decrease or even elimination of machining are some of the reasons for this. As a result, extruded materials surpass the original material in terms of mechanical qualities. [4]. The efficiency of the metal produced by backward extrusion process with an opposite direction to the punch movement is relatively higher than that when it moves in the same direction. This might be the result of friction between it and the chamber walls, which is avoidable. [5].

The importance of modeling and simulation has grown recently in the metal-forming sector, where it is seen as a crucial method for developing new products and processes. Process development in industrial extrusion is heavily reliant on trial and error, and full-scale experiments are frequently a part of this process. Many of these trials, which are typically costly and time-consuming, could be replaced by numerical simulations [1]. Based on the upper bound method, an analytical formulation solution for the forward extrusion of non-axisymmetric sections was created. This solution could be used to predict the material flow pattern in the forward extrusion process and greatly simplify the design of the die and the necessary tooling for a successful process. A DEFORM_3D commercial software was used in simulating the forward extrusion process. A punch velocity of 1mm/s was used in the FEM model. In FEM model the dimension of the model, constrains and simulation conditions were taken similar to those in the experiments process. Results obtained in the study, such as extrusion pressure and velocity field were found to be in a close agreement to the experimental and numerical ones [6]. Additionally, a simulation model was put up for the cold backward extrusion of copper cans. The model was run for a variety of punch-face forms, including concave, flat, and conical with conical angles of 900 and 1500. The investigation of effective strain and flow stress distributions in longitudinal sections of cold backward extruded copper cans was demonstrated by the obtained results. [5]. In addition, a three-dimension finite element simulation in the backward extrusion of an aluminum box-shaped was proposed using the ANSYS/ LS-DYNA finite element package. Researchers used the FEM to predict the forming force in the backward extrusion and the shape of the product at any stage of the forming process together with the friction force conditions. Authors obtained forming force behavior, which was in a good agreement with the experimental data [7]. Moreover, a backward cold extrusion through the punch with square cross-section was addressed. Two different workpiece materials and two different friction conditions were considered. FE numerical analysis was validated experimentally. The results confirm the feasibility of the process, which can be successfully combined with other extrusion processes [8].

II. FINITE-ELEMENT MODELLING:

Furthermore, a simulation model for the cold backward extrusion of copper cans was created. With conical angles of 900 and 1500, the model was run for a range of punch-face shapes, including concave, flat, and conical. According to the data obtained, effective strain and flow stress distributions were investigated in longitudinal sections of cold backward extruded copper cans. Backward extrusion procedures are difficult to simulate numerically mostly due to the significant deformation that occurs on the extruded material (in this case, aluminum) during the process [9]. As a result, simulation mesh distortion may happen, potentially producing imprecise and insufficient findings. For this reason, FE analysis needs to be done with caution and a thorough comprehension of the physical phenomena involved in backward extrusion processes [10]. In this study, FE simulations were conducted using the commercial finite element program ANSYS. In order to reduce processing time and increase calculation accuracy, a 3D ax-symmetric FE model was made for the backward extrusion process, and analysis was done.

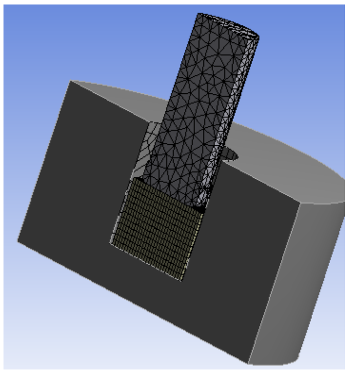


Fig. 1: Rigid, Die and Billet FE Mesh 3D-Symmetrical Models used for backward extrusion simulation

Only three components made up the FE analysis model: a punch, a billet of formed material, and a rigid die. The die was represented as the rigid body in the FE model, with no mesh produced. Figure 1 illustrates these three geometrical models.

The outer ring of the die was not modeled in order to simplify the numerical model. While displacement constraint was applied to the upper side of the aluminum billet, remote displacement was applied to the die's outside circumference (inner ring) as shown in Figure 2.

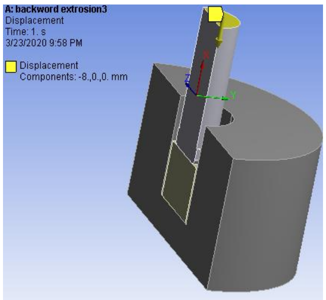


Fig. 2: Constraints used in the FE simulation

The die, as previously stated, was modeled as a rigid body. The die's stress and strain will not be examined in this manner. Therefore, the mesh was not formed, nor were the material qualities related to the die significant. By doing so, unnecessary computations that could result in a decrease in run time and numerical solution errors would be eliminated. The punch was made of hardened steel with strength higher than the aluminum billet. Therefore, stress and strain were not analyzed. The billet material (AL-6082) in the FE models have been modeled with Plane 183 finite element with 0.5 mm elements size. The Plane 183 is plastic and has a quadratic displacement. To replicate significant plastic strain deformation, multilinear isotropic hardening material was applied to aluminum billets using the ANSYS workbench [11]. The interface contacts between the billet-die and the billet-punch were represented as deformable. ANSYS handles these tasks using the contact-target surface technique, which ensures contact compatibility through an adjustable impenetrability constraint. The CONTA171 (surface to surface contact) was applied to a punch's surface at the point where the punch and aluminum billet met. Between billet-die interfaces, the TARGE174 element was used to simulate the additional surfaces at each interaction [10, 11]. It can be summarized that, in all interface contacts, the contact has been presented on the aluminum billet and the punch. On the other hand, the target has been presented on the inner circumference of the die. The element size was 0.3 mm, while the number of nods and used elements was 153900 and 37308 respectively. In this study, a commercial Al-Mg-Si alloy (Al-6082) was used. The stress-strain test data up to failure is necessary to describe the deformed material in simulation since the aluminum sample experiences a significant plastic-strain deformation during the forming process (Figure 3). As a multilinear isotropic hardening material, an aluminum billet was taken into consideration. von Mises yield criterion in conjunction with the isotropic work hardening assumption were applied in the FE simulations of aluminum billet behavior. The billet material's Poisson's ratio (v) is 0.334 and its elastic module (E) is 71GPa. It was expected that there would be no friction between the billet-die and billet-punch contacts.

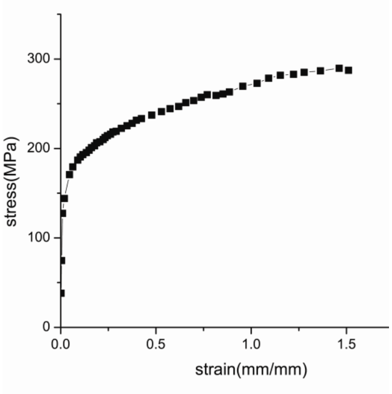


Fig. 3: Experimental tensile stress- strain curve for aluminum billet

III. EXPERIMENTAL WORK:

To test the manufactured material (Al-alloy), a backward extrusion technique was used on pieces that had an initial height of 10 mm and a diameter of 15 mm. The alloy's primary elements are Mg (0.6-0.12) %, Si (0.7-1.3) %, Mn (0.4-1) %, and Al 97%. To create cups with a 15 mm outer diameter and a 2 mm wall thickness, a stroke of 8 mm/min was used. Before it entered the die, the test sample was well lubricated. As seen in Figure 4, the die was mounted to the center of the TIRA test 2300 machine. The test was performed at room temperature. The equipment assembly drawing for the final step of the backward extrusion process is shown in Figure 5 [12].

Figure 6 demonstrates the workshop drawings of the active parts of the die set, punch, inner ring, and bottom insert, all machined in the department workshop, then exposed to quenching and tempering to get some toughness. The extruded aluminum cups produced in this experiment are demonstrated in Figure 7.

Fig. 4: The die set for backward extrusion

IV. RESULTS AND DISCUSSIONS

As previously stated, the forming load was applied as a displacement on the punch's upper side (Figure 2). Figure 8 illustrates the final 3D-symmetry step of FEM model of the proposed backward extrusion model.

The cup's equivalent stresses (in MPa, left) and plastic strains (in mm/mm, right) after the completion of the forming procedures for the 3D-symmetry FEM model are displayed in Figure 9. Figure 9 displays the highest and lowest values for stress and strain. Figure 10 shows the 3D-symmetry of the entire deformation of the FEM model for the final cup extrusion.

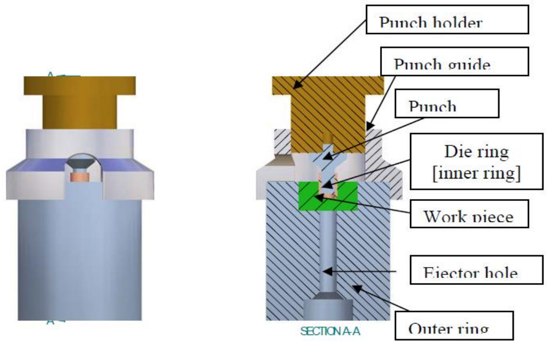


Fig. 5: CD Equipment for backward extrusion

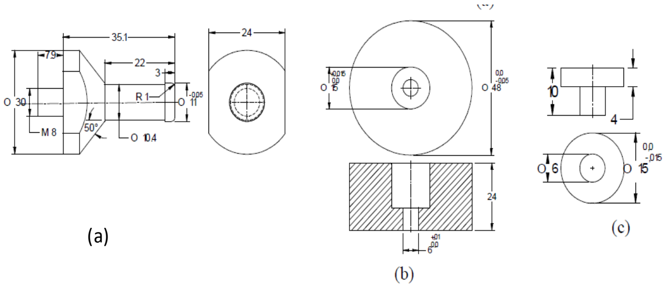


Fig. 6: The active parts of backward extrusion die: (a) punch, (b) inner ring (c) bottom insert (all dimensions in mm)


Fig.7: A view of the backward extruded cup

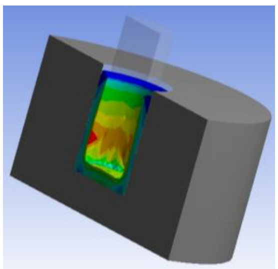


Fig.8: Final step of FEM model for backward extrusion

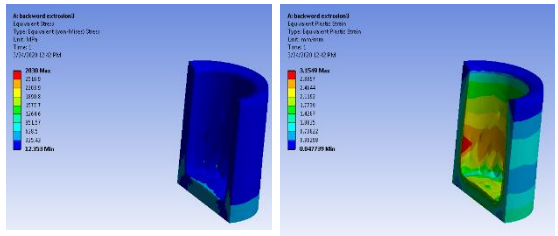


Fig. 9: Equivalent stress (left) and plastic strain (right) in extrude part

In order to validate the FE simulation results, the backward extrusion of an aluminum billet with a diameter of 15 mm and an initial height of 10 mm was experimentally performed. Figure 11 shows the load-ram travel diagram obtained from both the numerical simulation and the experiment. From Figure 11, it can be seen that the forming load curve from the FE simulation closely matches the experimental curve, indicating similarity between the experimental and numerical results. For further comparison, Figure 12 presents the final shape of the extruded cup produced by the FE simulation and the experiment. As shown, the final shapes of the cups (from both the FE model and the experiment) are very similar, suggesting that the FEM of backward extrusion is a reliable method for predicting material behavior in backward extrusion.

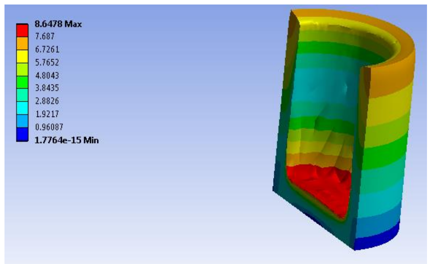


Fig. 10: Total deformation of cup simulation model.

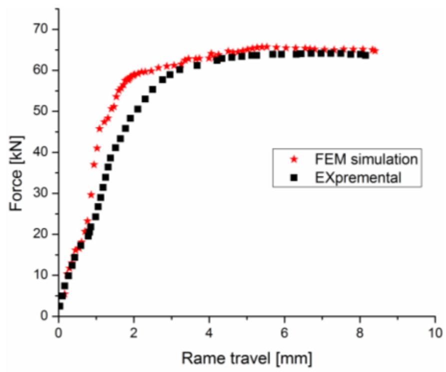
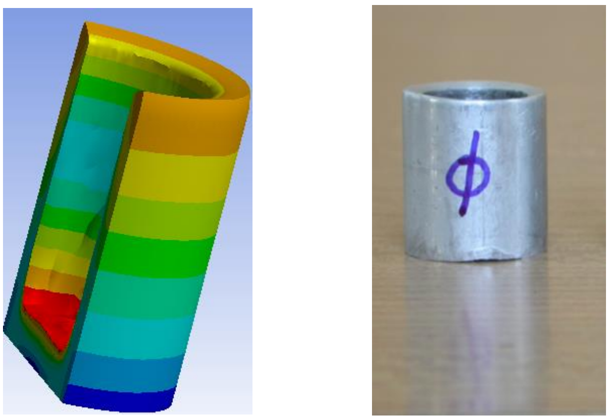


Fig.11: Backward extrusion force as a function of ram travel for Al 6082

Based on the results mentioned previously, the backward forming process finite element simulation may prove to be a valuable resource for comprehending and enhancing forming operations. Where it offers essential data for determining the forming parameters and operating duration. Also, the efficacy of the developed FE models and the proposed technique could be noticed in the predictions of the final product as well as the region of possible crack appearance.

V. CONCLUSION

The experimental and numerical investigation was successfully performed to predict the extrusion load of aluminum (AL-6082) alloy during the backward cup extrusion process. Experiment was carried out in order to validate the FEM simulation. Obtained results confirm the feasibility of the process, which could be successfully compared with other extrusion processes. Minor deviation between force values obtained experimentally and numerically can be attributed to the uncertainty in friction, flow stress determination, and the set-up of the numerical simulation model. High matching was noticed in the final shape of the extruded cup produced by the FEM simulation and the one produced by experiment techniques. For further research, authors will develop a simulation model for addressing the impact of some parameters such as friction, ram velocity, and temperature on the extrusion process in order to validate them experimentally for common materials used in such processes. Moreover, authors intent to study the validation of the backward extrusion simulation of the nanostructural material with their experimental results.



a b

Fig. 12: Backward extrusion for Al6082 (a) FEM simulation, (b) Experiment

ACKNOWLEDGEMENTS

This research did not receive any specific grant from funding agencies in the public, commercial, or not-for-profit sectors.

REFERENCES

[1] Kapadia, N. and Desai, A., 2015, Optimization Study of Die Extrusion Process using Finite Element Method (FEM), International Journal for Scientific Research & Development, 3, 9, 1007-1009.

[2] Peng, Q., Dong, H., Tian, Y. and Zhang, H., 2012, Effect of Backward Extrusion on Microstructure and Mechanical Properties of Mg-Gd Based Alloy, Materials Science and Engineering: A, 532, 443–448.

[3] Wang, Y. and Jiang, Z., 2012, Dynamic Compressive Behavior of Selected Aluminum Alloy at Low Temperature, Materials Science and Engineering: A, 553, 176–180.

[4] Hosseini, S. H., Abrinia, K. and Faraji, G., 2015, Applicability of a Modified Backward Extrusion Process on Commercially Pure Aluminum, 65, 521-528.

[5] Milek, T., 2017, The Analysis of Distributions of Effective Strain and Flow Stress in Longitudinal Sections of Cold Backward Extruded Copper Cans for Different Punch-face Shapes, II International Conference on Computational Methods in Engineering Science (CMES'17), Lublin, Poland.

[6] Nosrati A., Abrinia K. and Parvizi A., 2020, An Applied Analytical Method for the Forward Extrusion of Metals, Journal of Materials Engineering and Performance.

[7] Semenov I. and Ivanov A., 2020, Numerical simulation of backward extrusion process with local acting of active friction forces, IOP Conf. Series: Materials Science and Engineering 709 (2020) 022069.

[8] Plancak, M., Vilotic, D., Invanisevic, A., Movrin, D. and Krsulia, M., 2012, Backward Cold Extrusion of Aluminum and Steel Billets by Non-Circular Punch, Transfer inovácií, 22, 179-182.

[9] Benisa, M., Bojan, B., Grbovic, Al. and Stefanovic, Z. N., 2012, Computer-Aided Modeling of the Rubber-Pad Forming Process, Materials and technology, 46, 5.

[10] Benisa, M., Younise, B. and Aswihli, H., 2019, Experimental and Numerical Simulation for Bulk Nanostructure of AL-6082 Alloy Material Produced by Equal-Channel Angular Pressing (ECAP), The International Journal of Engineering and Information Technology (IJEIT), 6, 1.

[11] Benisa, M., Bojan, B., Grbovic, Al. and Stefanovic, Z. N., 2014, Numerical Simulation as a Tool for Optimizing Tool Geometry for Rubber Pad Forming Process, FME Transactions, 42, 67-73.

[12] Benisa, M., Eldabee, F. and Elderwish, A., 2019, The effects of Backward Extrusion of Bulk Nanostructured Al-6082 Materials Produced by Equal-Channel Angular Pressing (ECAP), Journal of Sciences and Technologies (Engineering and Applied Science), 3, 1, 52-65.