The Effect of Paraffinic Mineral Oil Lubricant in Cold Forward Extrusion

S.M. Hafis[1,*], M.J.M. Ridzuan[1], S. Syahrullail[2], A. Amran[2]

[1]Mechanical Engineering Programme, School of Mechatronic Engineering, Universiti Malaysia Perlis (UniMAP), 02600 Pauh, Perlis, Malaysia.
[2]Faculty of Mechanical Engineering, Universiti Teknologi Malaysia (UTM), 81310 UTM, Skudai, Johor, Malaysia.

Abstract: This paper presents the finite element analysis of cold forward extrusion and the analysis of the contact sliding behaviour on the die-billet surface by paraffinic mineral oil lubrication with kinematic viscosity of 92 mm2/s at 40 °C. The analysis dealt with the plasticity flow that was investigated by finite element method in order to identify the loads acting on the billet. The finite element analysis of stresses was performed based on load distributions calculated from experimental test. The time behaviour of displacements on the billet was then used as inputs for the extrusion model. The presented method provides good results with reduced computation time. The results of the extrusion model revealed that the zones of high stress located at the sharp edges of the die, which explains the observed extrusion force to reach a peak value.

Keywords: Extrusion, paraffinic mineral oil, finite element analysis, extrusion load.

1. Introduction

Paraffinic palm oils have been utilised in Malaysia for many purposes, for example, in metal forming process. A research was conducted [1] on the application of palm oil lubrication on tools to perform better surface finish on a product. The influence of palm oil lubricant viscosity on material deformation may differentiate the plasticity flow patterns within the extruded part and affects the necessary extrusion loads [2]. In addition, in order to determine whether a particular billet can be extruded to a given geometry without failure during the process, many studies [3, 4] have been conducted using numerical method to predict the stress and strain distributions.
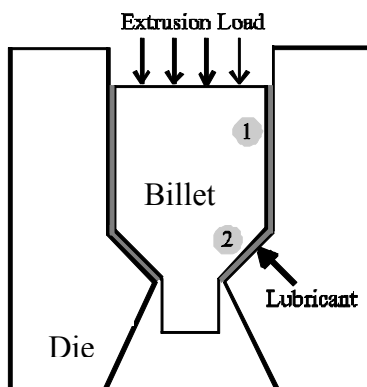

Fig. 1 Schematic diagrams of cold forward extrusion.

In cold forward extrusion operation, lubrication plays an important role in the two contact surface regions. Fig. 1 shows region 1 and region 2 that are located in billet-die container and billet-taper die surfaces, respectively. In this analysis, finite element analysis was performed to predict the effect of the palm oil lubricant.

2. Material Property and Friction Model

The material behaviour in the billet was used as an input for the finite element model. It was determined by the uniaxial tensile test on pure aluminium alloy AA1100. The test method followed ASTM E8M-91 standard. The relationship between true stress and true strain is presented in Fig. 2. The properties of the material were; density, ρ of 2700 kg/m^3, elastic modulus, E of 69 GPa, initial yield strength, σ_Y of 56 MPa, and Poisson's ratio, v of 0.33.
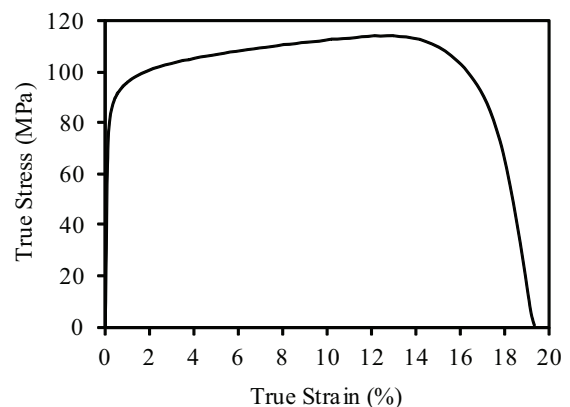

Fig. 2 The true stress-strain curve of aluminium alloy AA1100 by uniaxial tensile test.

It is reported that the lubricant quantity acts as friction coefficient between sliding surfaces [5]. In this case,

lubricant quantity was employed as friction coefficient because the different friction coefficient representing lubricant quantities between the die-billet surface. The friction law used to perform the simulation was Coulomb's law. It was chosen because high extrusion load condition can be applied when the friction coefficient on the die-billet contact surface is small [6].

3. Finite Element Method

Cold forward extrusion was modelled using two-dimension symmetrical half with 4-node elements using finite element software, Abaqus. A fine uniform mesh was applied in the billet to ensure accurate analysis. The billet was treated as elastic-plastic material. Fig. 3 shows the die-billet geometries were in accordance to the experimental work thus gave an extrusion ratio of 3, extrusion angle of 45°, and square billet with thickness of 9 mm. The punch and die were modelled by an analytical rigid surface because they were assumed to be rigid bodies and no deformation occurred during extrusion.
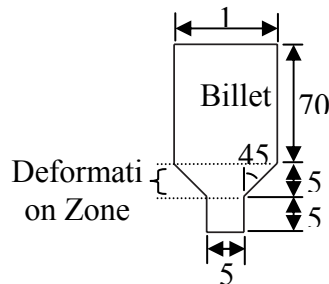

Fig. 3 Finite element model of cold forward extrusion (All dimensions in mm).

4. Validation of Finite Element Model

The cold forward extrusion simulation results must be reliable. The results obtained from the validation will ensure that the results of finite element analysis are accurate. To prove that the model is successful, comparison was made with the previous work by Gouveia et al. [7]. Gouveia et. al. stated that the billet had initial diameter of 15 mm and length of 45 mm. The diameter of the die was 12.55 mm and the angle of the extrusion die was 30°. The heights of the die container and the die opening were 56.88 mm and 1 mm, respectively. The material for this numerical simulation was mild steel 1015 with material behaviour i.e. $\sigma = c\varepsilon^n$, where σ is stress, ε is strain, n is strain-hardening constant and c is material constant. The material constant used were $c = 685.2$ MPa and $n = 0.185$. The properties of the material were; density, ρ of 7800 kg/m^3, elastic modulus, E of 207 GPa, Poisson's ratio, v of 0.3, and initial yield strength, σ_Y of 390 MPa. The friction coefficient, μ was 0.05 for all die-billet contact surfaces.
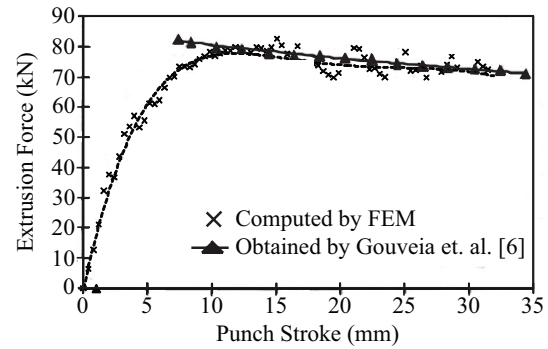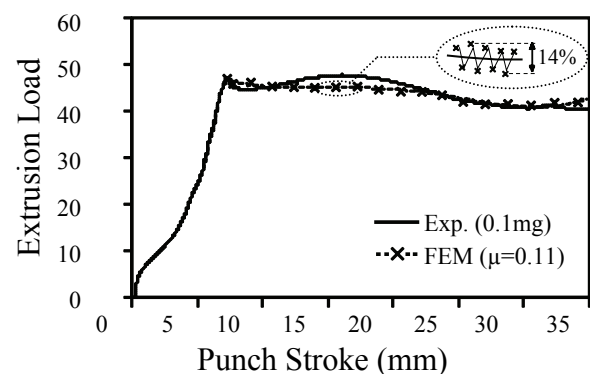

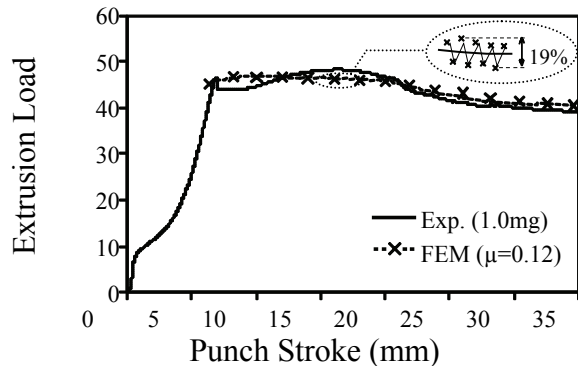Fig. 4 Validation of extrusion loads.

Fig. 4 shows that the extrusion load curve is in good agreement with the distribution loads by Gouveia et. al. [7]. This proves that the simulation result was valid and reliable. The oscillation error occurred due to computational errors of the repetitive numerical calculation in the finite element method. The errors can be overcome by refining the mesh elements in the billet. Since the finite element analysis of the previous work by Gouveia et. al. were performed and validated, finite element analysis of cold forward extrusion was executed. The result from this finite element analysis was confirmed to be reliable and behaved as in the actual process.
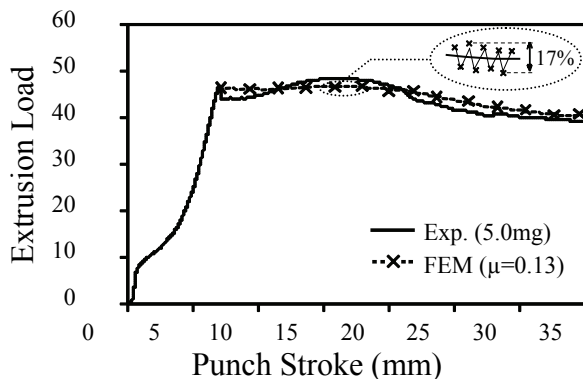
5. Results and Discussion

In the attempt to find the friction coefficient for the increasing lubricant quantity, the extrusion load distributions imposed on top of the billet were analysed. The time behaviour of displacements obtained from the experimental results was used as the inputs in the finite element analysis and the friction coefficient was determined by heuristic solution.

(a)

(b)

(c)

Fig. 5 Comparison of extrusion load for finite element analysis and experimental results with lubricant quantity in amount of (a) 0.1 mg, (b) 1.0 mg, and (c) 5.0 mg.

The extrusion load distributions in the steady-state condition of the finite element method were analyzed and compared with the experiment results. From Fig. 5, the extrusion load using finite element analysis present undesirable oscillations. The finite element results plotted are the best polynomial fit to the finite element data, the deviation was found to be in a range of 14%and 19%. Fig. 5(a) shows friction coefficient of 0.11 fitted to the experimental results with the quantity of paraffinic oil of 0.1 mg. Figs. 5(b) and 5(c) demonstrate extrusion load distributions with friction coefficient of 0.12 and 0.13 for 1.0 mg and 5.0 mg of paraffinic oil quantities, respectively. It is worth mentioning that the extrusion load curves for paraffinic mineral oil lubricant quantity in amount of 0.1 mg, 1.0 mg and 5.0 mg were in good agreement with the predicted friction coefficient within 0.11 to 0.13. These conditions can be explained by the rate changes of plastic deformation in the billet and the contact force along the die wall. The die geometry and extrusion ratio may contribute to these results as well [6].
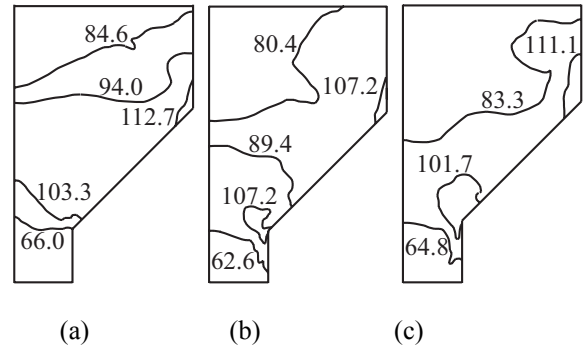

Fig. 6 The von Misses stress patterns for friction coefficient, (a) μ = 0.11, (b) μ = 0.12, and (c) μ = 0.13 (all dimensions are in MPa).

Fig. 6 shows the stress distributions in the billet during extrusion process once the steady-state was achieved. In this finite element analysis, the von Misses stresses were readily derived from Abaqus software. The stresses were greater once the billet started to flow through the die opening and this led to the occurrence of stress concentrations at the curve die edges that required high extrusion loads. However, the fracture deformation in the billet could occur if the stress concentrations were too large.

Fig. 7 shows the mesh flow pattern for three different finite element extrusions. The figure presents the differences in friction coefficients that affected the deforming mesh flow patterns right after die opening. The simulations utilised Arbitrary Lagrangian-Eulerian (ALE) mesh formulation. This made the finite element mesh moved through space with the deforming elastic plastic material. As a result, the changes in mesh flow of the product can be followed easily.
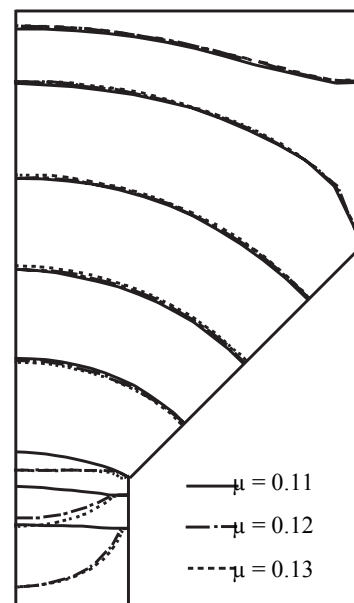

Fig. 7 The mesh flow pattern at the deformation zone.

6. Conclusion

A successful finite element method for paraffinic palm oil lubrication on the die-billet contact surface has been discussed. The extrusion load distributions were successfully evaluated based on lubricant quantities and friction coefficients on the die-billet interfaces. The finite element model was reliable once a good agreement with the previous work was found thus it confirmed the accuracy of the finite element model. The friction coefficient obtained for the three different paraffinic mineral palm oil lubricant quantities was observed in a range of 0.11 to 0.13. The analysis has shown that the influence of the lubricant quantity on the stress distribution in extruded specimens is large at the exit region of the deformed zone, particularly close to the taper die contact surface. The finite element analysis performed concluded that the higher the friction coefficient, the greater the extrusion loads.

References

[1] Syahrullail, S., Azwadi, C.S.N., Seah, W.B., (2009), Plasticity analysis of pure aluminium extruded with an RBD palm olein lubricant. Journal of Applied Sciences, 9 (19), 2009, p. 3583-3585.

[2] Syahrullail, S., Azwadi, C.S.N., Ridzuan, M.J.M. , Seah, W.B., (2009), The effect of lubricant viscocity in cold forward plane strain extrusion test. European Journal of Scientific Research, 38 (4), p. 550-551.

[3] D.Y. Yang, C.M. Lee, J.H. Yoon, (1989), Finite element analysis of steady-state three-dimensional extrusion of sections through curve dies. International Journal Mechanical Science, 31 (2), p. 151-154.

[4] L. H. You, J. H. Hu, Y. H. Shi, J. J. Zhang, (2004), Single-patch surfaces for tool shape design and finite element analysis of hot extrusion. Journal of Material Processing Technology, 150 (1-2), p. 65-68.

[5] Z. M. Hu, T. A. Dean, (2000), A study of surface topography, friction and lubricants in metal forming, International Journal of Machine Tools and Manufacture, 40 (11), p. 1637-1647.

[6] Zhang, Q., Arentoft, M., Bruschi, S., Dubar, L., Felder, E., (2008), Measurement of friction in a cold extrusion operation: Study by numerical simulation of four friction tests. International Journal Material Forming, 1 (1), p. 1268-1270.

[7] B.P.P.A. Gouveia, J.M.C. Rodrigues, N. Bay, P.A.F. Martins, (1999), Finite element modelling of cold forward extrusion. Journal of Material Processing Technology, 94 (2-3), p. 88-91.