The deformation of a flat die in aluminum extrusion

W.Assaad, H.J.M.Geijselaers, J.Huétink

Faculty of Engineering Technology, University of Twente
P.O.Box 217, 7500 AE Enschede, The Netherlands
phone: +31-(0)53-4894069, email: w.assaad@ctw.utwente.nl

Introduction

In a direct aluminum extrusion process the die is subjected to mechanical and thermal loads. These loads cause the die face to deform in a concave shape. A product with unacceptable specifications is produced when the deformation is too big. 3D finite element simulations are required to give a better understanding of aluminum flow and die deformation. These simulations help in designing the suitable die. But these simulations are faced with one of several problems that must be overcome such as calculation time. As an example, an isothermal finite element simulation of the extrusion of a U-shape profile is performed and the deformation of the die is determined. These numerical results will be validated experimentally in the coming period.

Numerically

The decoupled analysis is applied in evaluating the deformation of the die. Eulerian and Updated lagrangian and formulations are applied for the material and the die respectively. Viscoplastic and elastoplastic material models are applied for aluminum and die respectively. The model is discretized with 10-node tetrahedron element.

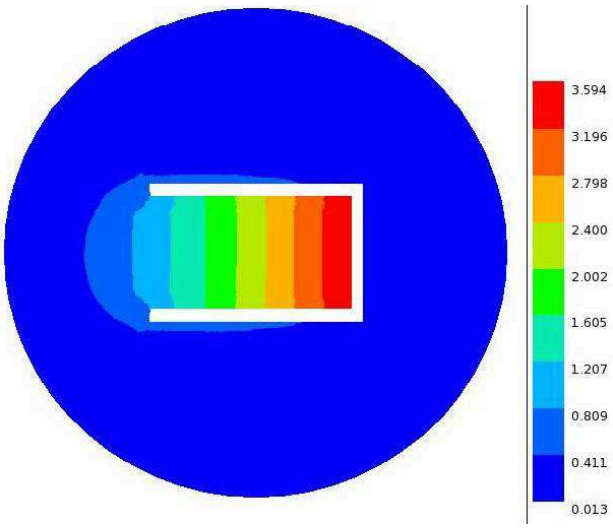

Figure 1 : Die deformation(mm) in extrusion direction

Experimentally

To measure the die deformation a laser beam is applied on a stainless steel flat mirror which is fixed on the die. When the die deforms, the reflected beam deviates. The deformation of the die and the deviation of the reflected beam are related linearly through geometrical relations.

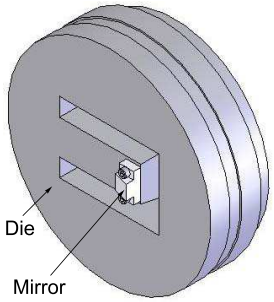

Figure 2 : Die with the stainless steel mirror

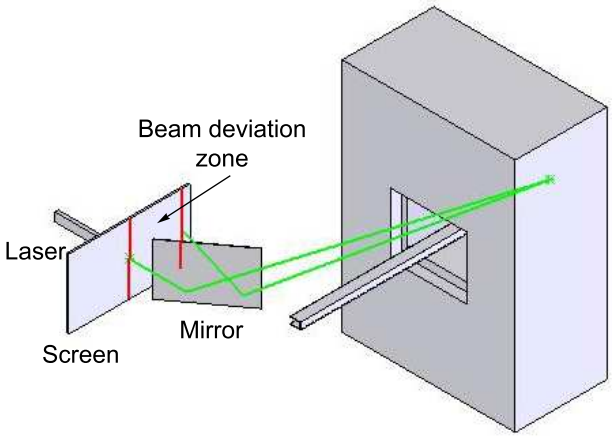

Figure 3 : Experiment setup

Future Work

Modifications on the decoupled analysis will be considered to increase its accuracy and simplicity. In addition, a thermo-mechanical simulation will be performed as soon as a thermo-mechanical tetrahedron element will be implemented.