

Modelling and simulation of hollow profile aluminium extruded product

ABSTRACT

The main objectives of this paper is to find the way for solving the problems of aluminum extrusion process, and improve the mechanical properties of the products through a smart design, modelling and simulation of this process by using finite element method (FEM). For the purpose to model a (2D) two dimensions warm aluminum extrusion process, ABAQUS software was used to set up the finite element simulation. The main parameters which have major effects on this process like extrusion stresses, temperature, and die geometry, i.e. extrusion radius, were taken into consideration. Aluminum alloy (Al-2014) was used as the billet material, with 40 mm diameter and 75 mm length. It is important to preheat the billet from the beginning to a specific temperature, and then pressurizes it into the die. This process is an isothermal process with an extrusion ratio of 3.3. Subsequently, the optimized algorithm for these extrusion parameters was suggested based on the simulation results. The results suggest that the large die angle needs a less extrusion load than the small die angle. In all die geometry used, the deformation of aluminum billet, which caused by shearing and compression stresses, happened in a small sectional area, i.e., bearing area. The results also showed that the values of these stresses can increase or decrease depends on the die entrance angle and the die bearing length. To avoid the effects of these stresses on die dimensions; the hardness, material selection, and geometry should be well calculated.

Keyword: Aluminium alloy; Aluminum extrusion; Analysis; Extrusion process; FEM simulation