Abstract

This study employs Finite Element Analysis (FEA) using ABAQUS/CAE to investigate the hot direct extrusion process, with a focus on deformation mechanics, applied forces, and the interplay of critical parameters such as temperature, die angles, profile radius, coefficient of friction , and material properties. The research bridges analytical methodologies from manufacturing theory with advanced numerical simulations to validate the predictive accuracy of FEA in industrial applications. A comparative analysis of AA6061 aluminum and Ti-6Al-4V titanium alloys underscores the influence of material behavior on extrusion forces, product integrity, and residual material retention.

A comprehensive field visit to NAPCO (National Aluminum & Profile Company) in Palestine provided empirical insights into industrial extrusion workflows, including billet preparation, die design, and post-processing treatments. Numerical simulations explored the effects of die geometry (semi-die angles, fillet radii) and interfacial friction on stress distribution, temperature gradients, and deformation homogeneity. Results revealed that elevated temperatures reduce flow stress in AA6061, while optimal die angles (45°–60°) and increased fillet radii (10–20 mm) mitigate contact pressure by up to 67%, enhancing material flow efficiency. For Ti-6Al-4V, strain rate sensitivity was pronounced, with higher rates amplifying stress concentrations near die interfaces.

This work validates FEA outcomes against theoretical models and industrial benchmarks, offering actionable strategies to optimize extrusion parameters, minimize operational costs, and improve product performance. By integrating computational rigor with practical manufacturing insights, the study advances the application of FEA in metal-forming industries.