Application of Computational Fluid Dynamics (CFD) Simulations to Spray-Freezing Operations

C. Anandharamakrishnan[a], J. Gimbun[ab], A. G. F. Stapley[a] & C. D. Rielly[a]*
[a]Department of Chemical Engineering , Loughborough University , Loughborough, Leicestershire, United Kingdom
[b]Faculty of Chemical and natural Resources Engineering, University Malaysia Pahang, 26300, Gambang, Kuantan, Pahang, Malaysia

ABSTRACT
A 3D computational fluid dynamics (CFD) simulation for spray-freezing in a cold gas has been developed and used to identify design improvements. This model includes an approximate method to model the latent heat of fusion and is able to track particle trajectories. The simulation predictions agreed reasonably well with experimentally measured gas temperatures and droplet velocities. The results suggest that a hollow-cone spray is more effective in cooling the particles uniformly. The CFD simulation suggested that buildup of an icy layer on the cone walls observed experimentally was due to incomplete freezing of larger particles (>100 μm). Collection efficiencies could be raised (from 20 to 57%) by increasing the diameter of the chamber outlet.

KEYWORDS: Impact position, Recalescence, Residence time, Solidification

DOI: 10.1080/07373930903430843