Contribution ID: **418** Type: **Oral Presentation**

An OpenFOAM Eulerian-Lagrangian Model of a Polydisperse Bubbly Flow in a Continuous Casting Mould

Wednesday 8 October 2025 17:50 (20 minutes)

Some defects in continuous cast (CC) steel are the result of flow conditions at the top section of the CC mould. To monitor and potentially control the flow, in order to reduce such defects, information about the flow conditions is needed. In addition to sensors that may be installed on the mould, CFD can provide detailed insight into the mould flow. We are developing an OpenFOAM-based Eulerian-Lagrangian multiphase flow solver for such simulations, based on an existing OpenFOAM Lagrangian solver, DPMFoam. One limitation of that solver is that the size of particles or bubbles be much smaller than the mesh size, in order to produce accurate results; this constraint limits the use of mesh refinement in regions of interest, and hinders obtaining grid independent results. Another limitation of the Lagrangian method is that the free surface between water and air atop the mould is not modeled. In this work, a multi-stage diffusion method has been added to the OpenFOAM solver DPMFoam, to resolve the restriction of particle to mesh size; and a Volume of Fluid method has been added to the solver to model the water/air free surface. Simulation results of polydisperse bubbly flows in a CC mould will be presented, and those results compared to corresponding experimental data from a full scale water model of the top of a CC mould, focusing on the behaviour of bubbles as a function of size, and of fluctuations of the water/air free surface, as a function of various casting conditions.

Primary authors: Prof. BUSSMANN, Markus (Department of Mechanical & Industrial Engineering at the University of Toronto); AMIRZADEH, Saeid

Presenter: AMIRZADEH, Saeid

Session Classification: Digitalization, AI & Modeling

Track Classification: Steelmaking - Continuous casting, near-net shape casting and ingot casting