6th European Conference on Computational Mechanics (ECCM 6)
7th European Conference on Computational Fluid Dynamics (ECFD 7)
11-15 June 2018, Glasgow, UK

A ROBUST PARTICLE–BASED SOLVER FOR MODELING HEAT TRANSFER IN MULTIPHASE FLOWS

M. AFRASIABI[1,2,*], M. ROETHLIN[2], E. CHATZI[1] AND K. WEGENER[2]

[1] Institute of Structural Engineering (IBK), D-BAUG, ETH Zürich
Stefano-Franscini-Platz 5, 8093 Zürich, Switzerland
[2] Institute of Machine Tools & Manufacturing (IWF), D-MAVT, ETH Zürich
Leonhardstrasse 21, 8092 Zürich, Switzerland
E-mail: afrasiabi@ethz.ch

Keywords: *Multiphase flows, Heat transfer, Interfaces, Meshfree methods, Level Set.*

Designing a robust numerical software for simulating heat and mass transfer is of paramount importance in fluid dynamics problems, according to its countless contributions in literature. Among a broad array of numerical techniques in such application areas, meshfree particle–based methods stand one of the most suitable choices due to their unique strength in handling large deformations and high gradients. There is, nevertheless, no work dedicated in this context to the use of the interconnected state-of-the-art meshfree techniques in multi-physically coupled problems, where a number of phenomena such as multiphase flows, heat transfer, and moving interfaces are encountered.

To fill this demanding gap, the present article introduces a novel coupling by bringing together a set of three contemporary particle–based methods from the state of the art, capable of simulating heat transfer in multiphase flows with minimum error. The idea originates from increasing the robustness of the geometrical features required to calculate physical quantities, such as the boundary conditions in the heat equation and the surface tension force in Navier-Stokes equations. As a result of this improvement, a fully second-order approximation of the overall solution is achieved through the enhancement of each constitutive subset, motivated by the accuracy vs. efficiency trade-off.

The workability of this novel approach for handling thermal effects in complex moving interfaces is examined with a numerical test case wherein multiple scenarios such as internal flow, bursting at free surface as well as bubble evolutionary motions occur. Performing a benchmark comparison with the reference model implemented in `COMSOL Multiphysics®`, a markedly improved mass conservation using the proposed particle–based solver is observed. This, in turn, results in a more realistic and accurate approximation of the temperature. The proposed numerical approach is shown to achieve efficiency previously impossible for the same resolution, creating a vast potential for the simulation of many industrial CFD engineering applications where the accuracy is a major concern. [1]

[1]Readers are strongly encouraged to refer to the full paper of the present work, to be published in the same proceedings.