14th World Congress on Computational Mechanics (WCCM XIV)
8th European Congress on Computational Methods in Applied Science and Engineering (ECCOMAS 2020)
July 19- 24, 2020, Paris, France

INTERFACIAL FLOW SIMULATION

M. S. SHADLOO*, A. RAHMAT† AND M. YILDIZ‡

* Normandiy University, Coria Lab. CNRS-UMR6614, University and INSA or Rouen, 76000 France
E-mail: msshadloo@coria.fr

† School of Chemical Engineering, University of Birmingham, B15 2TT, Birmingham, United Kingdom
E-mail: a.rahmat@bham.ac.uk

‡ Integrated Manufacturing Technologies Research and Application Center, Faculty of Engineering and Natural Sciences, Sabanci University, Tuzla, 34956 Istanbul, Turkey
E-mail: meyildiz@sabanciuniv.edu, myildiz@uvic.ca

Key words: Computational Fluid Dynamics (CFD), Multiphase flow, Free surface flow, Fluid structure interaction, Numerical methodology

ABSTRACT

This mini symposium (MS) is dedicated to interfacial flow simulation (multiphase flows, free surface flows and fluid-structure interaction). One of the main goals of this MS is to gain an insight into the computational aspects of interfacial flow simulations by considering five following distinguished common parts of each numerical technique, i.e. (i) flow modeling, (ii) interface treatment, (iii) flow-interface coupling, (iv) spatial discretization schemes, and (v) the flow equation solver. We concisely invite the following manuscripts: the use of conventional numerical methods such as Finite Difference (FDM), Finite volume (FVM) and Finite Element (FEM) methods for the simulation of interfacial flows and to elaborate on their differences, similarities, advantages and drawbacks. As such, the development, validation and benchmarking of less established and newly attracting numerical methodologies such as Smoothed Particle Hydrodynamics (SPH), Moving Particle semi-Implicit (MPS), Lattice Boltzmann (LBM) methods, etc. are also in the core scope of this research topic.

Papers ranging from new physical modeling and discoveries to the correct treatment of difficulties inherent to numerical modeling of fluid flow system are invited for submission. These include but not limited to: (i) the correctly and effectively models physical boundary conditions; (ii) mass and energy conservations (iii) realistically treating the complicated physical interfacial phenomena such as folding, merging and/or break-up; (iv) properly taking the interfacial jump condition into account (i.e. large density and viscosity ratios in multi-phase and/or compressible flows); (v) extendibility to dealing with more complicated phenomena such as those in Magneto-hydrodynamics (MHD), Electro-hydrodynamics (EHD), non-Newtonian flows, phase change, nano-fluidic, etc. problems; and finally (vi) the extension of aforementioned methodologies to three-dimensional modeling and massively parallel computing in order to handle the real life problems are in particular interest.

We would be delighted if you could share one of your recent works with us and other interested colleagues in the community.