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

INTERFACIAL FLUID FLOW SIMULATION

MOSTAFA SAFDARI SHADLOO*, MEHMET YILDIZ†

* CNRS-University & INSA de Rouen
76000 Rouen, France
msshadloo@coria.fr

† FENS Department, Sabanci University
34956, Istanbul, Turkey
meyildiz@sabanciuniv.edu

Key words: Interfacial Flows, Flow-Interface Coupling, Multiphysics Problems, Computational Fluid Dynamics.

ABSTRACT

Interfacial flows (used in general sense to specify all kind of interfacial phenomena such as free surface flows, multi-phase flows, fluid-structure interaction, etc.) are of the most challenging and difficult to represent by CFD, whereas they are present in many industrial problems such as, e.g., cavitation on blades, boiling heat transfer in industrial processes, oil-water separation, wave energy converters, air entrainment at ocean surfaces, bubble reactors and tire grooves among others. Nevertheless, because of the complexity of these problems mainly associated with the necessity of accurately describing complex interface evolutions, most of the literature discusses only of simple problems. As can be inferred, the interface evolution is crucial to the modeling of such flows and thus, needs to be modeled correctly and studiously in order to obtain reliable simulation results. The ongoing attempts of modeling interfacial fluid flows resulted in the availability of a numerous amount of papers with different numerical approaches.

One of the main of this mini-symposium (MS) is to gain an insight into the computational aspects of interfacial flow simulations by considering the 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, (v) and the flow equation solver. We concisely invite the following manuscripts: the use of the 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 and validation of less established and newly attracting numerical methodologies such as Smoothed Particle Hydrodynamics (SPH), Moving Particle semi-Implicit (MPS), Lattice Boltzmann (LBM) methods 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 the 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 Magnetohydrodynamics (MHD), Electrohydrodynamics (EHD), non-Newtonian flows, phase change, nano-fluidic, etc. problems; and finally (vi) the extension of before mentioned methodologies to three-dimensional modeling and massively parallel computing in order to handle the real life problems are in particular interest.