Numerical simulation of two-phase fluid flows

Karthik Ramaswamy and Sadvik Shangerganesh
Department of Computational and Data Sciences,
Indian Institute of Science, Bangalore, India-560 012
E-mail: karthik@iisc.ac.in, shanger@iisc.ac.in

Use this template to prepare two-page (max) abstract for "Indo-German conference on Computational Mathematics (IGCM-2019): Challenges and Opportunities towards Exascale Computing".

Keywords: Finite element method, Fluid Flows

Section I

Numerical simulation of two-phase flows are important in industrial and biological applications. This work is concerned with the robust and efficient numerical simulation of incompressible two phase flows. In such flow problems, often, the evolution of the interface is induced by the surface force which consists of surface tension and the local curvature of the interface. Thus, an accurate representation of the interface is important in the numerical simulations to calculate the local curvature and include the surface force. Furthermore, two immiscible fluids will have different densities and viscosities, in general and therefore these physical variables will be discontinuous across the interface. It turns out that the development of robust and efficient numerical schemes for simulating fluid flows with unsteady motion of moving interfaces is still a challenging problem in computational fluid dynamics (CFD) field.

$$\frac{\partial \mathbf{u}}{\partial t} + \mathbf{u} \cdot \nabla \mathbf{u} - \frac{1}{\mathrm{Re}} \nabla \cdot \mathbb{D}(\mathbf{u}) + \nabla p = 0$$
$$\nabla \cdot \mathbf{u} = 0$$

The core of our methodology is the use of interface resolving meshes and the arbitrary Lagrangian-Eulerian (ALE) description of the fluid kinematics. Our numerical scheme is based on second order finite elements, a fractional step θ time discretisation, and a special approximation of the curvature to incorporate surface tension effects. We demonstrate the potential of the proposed numerical method by the simulation of a rising bubble and the Rayleigh-Taylor instability problem.

Section II

Numerical simulation of two-phase flows are important in industrial and biological applications. This work is concerned with the robust and efficient numerical simulation of incompressible two phase flows. In such flow problems, often, the evolution of the interface is induced by the surface force which consists of surface tension and the local curvature of the interface. Thus, an accurate representation of the interface is important in the numerical simulations to calculate the local curvature and include the surface

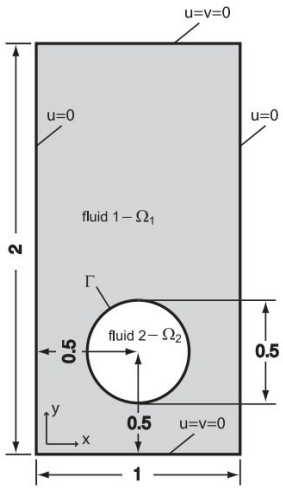

Figure 1: Template for IGCM 2019

force. Furthermore, two immiscible fluids will have different densities and viscosities, in general and therefore these physical variables will be discontinuous across the interface. It turns out that the development of robust and efficient numerical schemes for simulating fluid flows with unsteady motion of moving interfaces is still a challenging problem in computational fluid dynamics (CFD) field.

References

[1] S. Ganesan, V. John, G. Matthies, R. Meesala, S. Abdus, U. Wilbrandt, An object oriented parallel finite element scheme for computations of PDEs: Design and implementation, IEEE 23rd International Conference on High Performance Computing Workshops (HiPCW), (2016), 2-11.