Hybrid CFD simulation of two phase flow in inline flow splitters using VoF and Lagrangian models

Hanane Atmani[a*], Dominique Legendre[a], Rémi Zamansky[a], Eric Climent[a], Annaïg Pedrono[a]
Benjamin Sahovic[b], Eckhard Schleicher[b], Uwe Hampel[b],
Muhammad Awais Sattar[c], Laurent Babout[c], Robert Banasiak[c]

[a]Institut de Mécanique des Fluides de Toulouse (IMFT), Université de Toulouse, CNRS, Toulouse, France

2 Allée du Professeur Camille Soula, Toulouse, 31400, France.

[b]Helmholtz-Zentrum Dresden - Rossendorf, Institute of Fluid Dynamics, Dresden, Germany

[c]Lodz University of Technology, Institute of applied computer science, Lodz, Poland

* hanane.atmani@imft.fr

Keywords: CFD, two-phase flow, Inline fluid separation, Swirl, JADIM, VoF, IBM, Lagrangian tracking.

Abstract

Inline fluid separation using a swirl element is a recent technology for oil/gas extraction. Centrifugal forces up to 100 times the gravitational acceleration separate the phases, leaving the heavy phase next to the wall and the light one in the center. The current study, part of a european project TOMOCON, aims at developing CFD methods in the in-house IMFT code JADIM to simulate the two phase flow separation. Since the scales are ranging from 1 m (the characteristic length of the device :pipe, swirl element) to 10^{-6}m which can be the size of the smallest bubbles and drops, choosing the adequate approach between Euler/Euler and Euler/Lagrange models to simulate the inline fluid separation is one of the crucial parts of the work.
As the Euler/Euler method cannot describe precisely the interface and the bubble/bubble and bubble/fluid interaction which is one of the important parameters to evaluate the efficiency of the separation, a hybrid Euler/Lagrange approach has been selected consisting on the following strategy: First both the pipe and the swirl element are simulated using Immersed Boundary Method (IBM) for solid/fluid interaction, then the Lagrangian solver is used to track the dispersed phase (droplets/bubbles). Once the separation is done and the coalescence takes place leading to large volume of gas/oil compared to the mesh size, we switch to the Volume of Fluid (VoF) method to simulate the core inside the heavy phase. The following Navier Stokes equation summerizes the numerical schemes:

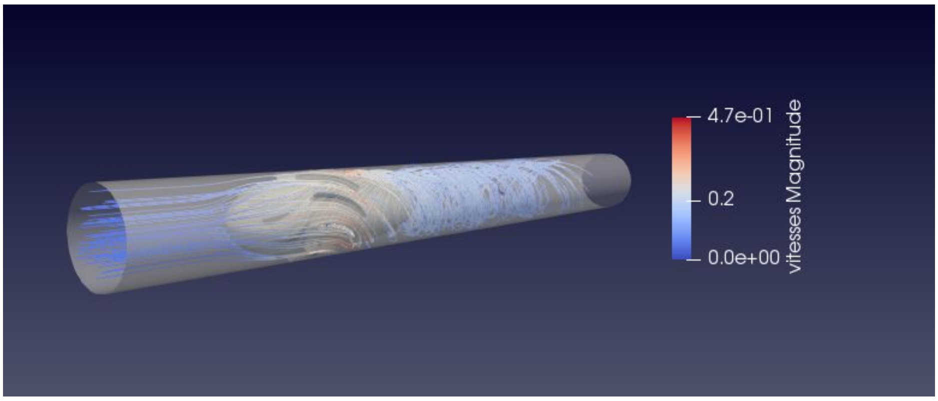
$$\rho(\frac{\partial \mathbf{u}}{\partial t} + \mathbf{u}.\nabla\mathbf{u}) = \rho\mathbf{g} - \nabla P + \nabla.\Sigma + \mathbf{f}_{IBM} + \mathbf{f}_{\gamma} + \mathbf{f}_{LAG} \tag{1}$$

with $\mathbf{u}$ the fluid velocity, P the pressure, $\mathbf{g}$ the gravity, Σ the viscous stress tensor. $\mathbf{f}_{IBM}$, $\mathbf{f}_{\gamma}$, $\mathbf{f}_{LAG}$ are the volumetric forcing coming from the IBM, VoF and Lagrangian methods respectively.
• *The Immersed Boundary Method (IBM):* first introduced by Peskin[1] for the simulation of fluid structure interactions is used here for both the pipe and the swirl element description. It consists in penalizing the fluid by introducing a volumetric force $\mathbf{f}_{IBM}$. In JADIM a solid volume fraction α is introduced which enables us to reduce the cost of computation and simulate different fixed and moving obstacles[2].
• *The Volume of Fluid (VoF):* first introduced by Hirt and Nichols[3] for interface tracking, it consists in tracking the interface through the VoF function C (or volume fraction) which equals to 1 in one phase and 0 in the other. The change of C is governed by the transport equation:

$$\frac{\partial C}{\partial t} + \mathbf{u}.\nabla C = 0 \tag{2}$$

The VoF solver in JADIM do not need any interface reconstruction and is based on an FCT solver. The capillary force $\mathbf{f}_{\gamma}$ in such approach requires a specific treatment to avoid spurious current[4].
• *The Lagrangian tracking:* is used for the description of bubbles having a diameter smaller than the mesh size, the bubbles are then considered as point force. The two-way coupling Lagrangian solver in JADIM calculates the trajectory of each bubble based on the forces applied on it (Drag, lift, buoyancy, Added mass, Tchen) and the feedback of the presence of the bubble is considered in both the mass and momentum equations (e.g. $\mathbf{f}_{LAG}$) for the fluid.
As mentionned before the hybrid CFD method will allow us to switch from the lagrangian tracking of the dispersed phase to a volume of fluid scheme after the core formation leading to a less computation cost.

A first illustration of the numerical simulation of a single phase flow through the separator is shown in the next figure. As expected, the velocity is found to be maximum next to the blades of the swirl element and decreases while the flow is getting far from it.



Example of streamlines of a single phase flow through the separator.

Numerical simulations of this industrial process will surely help fixing the physical parameters which influence the separation and control the efficiency. A comparaison with the experimental results from both HZDR using wire mesh sensor and TUL using electrical tomography will enable validating the models and build a control system for the inline fluid separation.

Acknowledgement:
This project has received funding from the European Union's Horizon 2020 research and innovation programme under the Marie Sklodowska Curie grant agreement No 764902.

References:
[1] C.S. Peskin. Numerical analysis of blood flow in the heart. J. Comput. Phys., 25(3), 220-252 (1977).
[2] B. Bigot, T. Bonometti, O. Thual, L. Lacaze, A simple immersed boundary method for solid fluid interaction in constant and stratified density flows. J. Computers and fluids 97:126142 (2014).
[3] C. W. Hirt, B. D. Nichols. Volume of Fluid (VoF) method for the dynamics of free boundaries. Journal of Computational Physics 39, 201-225 (1981).
[4] T. Abadie, J. Aubin, D. Legendre. On the combined effects of surface tension force calculation and interface advection on spurious currents within Volume of Fluid and Level Set frameworks. Journal of Computational Physics 297, 611-636 (2015).