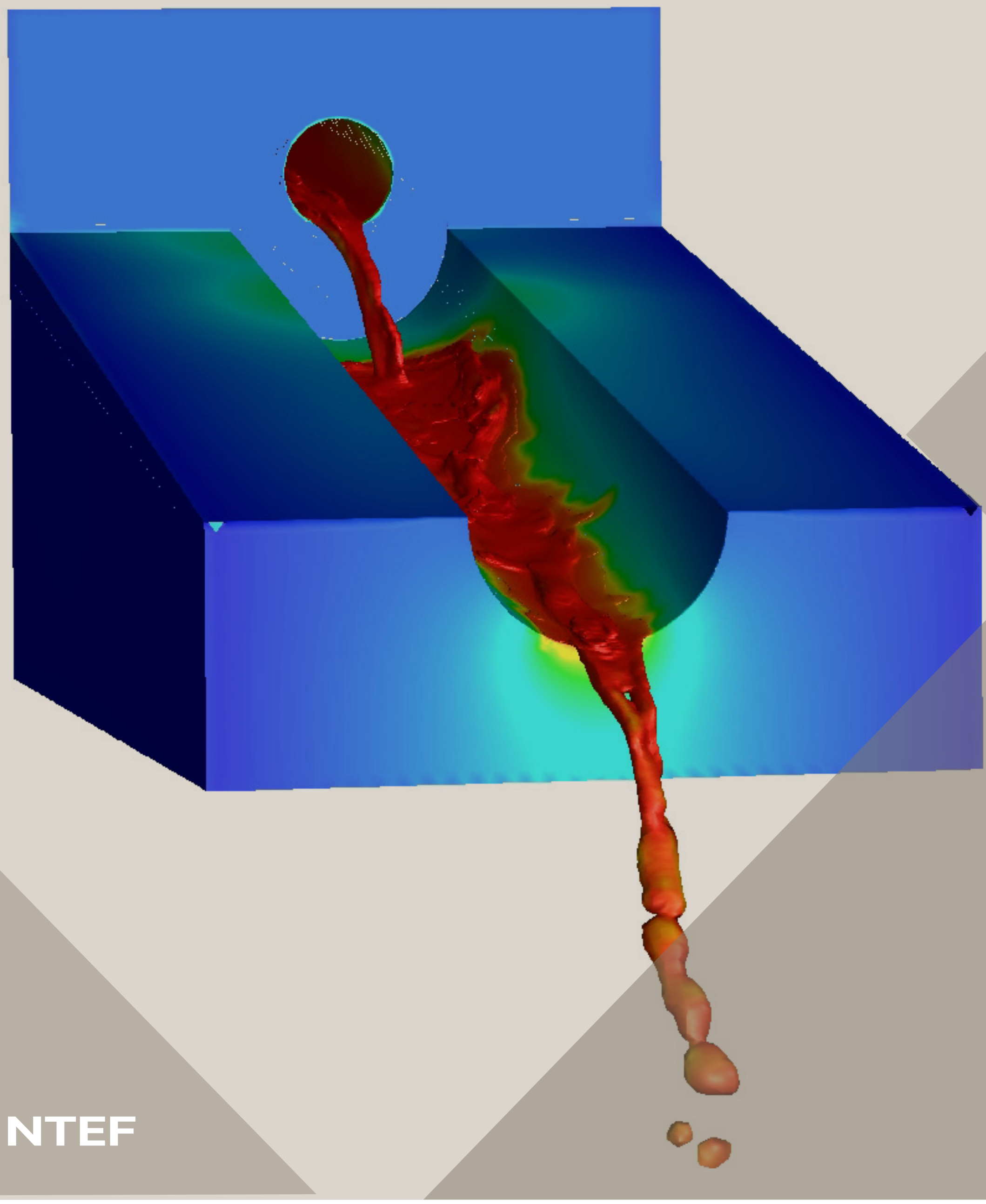
SINTEF
PROCEEDINGS
6

14th International Conference on CFD in
Oil & Gas, Metallurgical and Process Industries
SINTEF, Trondheim, Norway, October 12–14, 2020

Proceedings from the 14th International Conference on CFD in Oil & Gas, Metallurgical and Process Industries



SINTEF

SINTEF Proceedings

Editors:
Jan Erik Olsen, Jan Hendrik Cloete and Stein Tore Johansen

Proceedings from the 14th International Conference on CFD in Oil & Gas, Metallurgical and Process Industries

SINTEF, Trondheim, Norway
October 12–14, 2020

SINTEF Academic Press

SINTEF Proceedings 6
Editors: Jan Erik Olsen, Jan Hendrik Cloete and Stein Tore Johansen

Proceedings from the 14th International Conference on CFD in Oil & Gas, Metallurgical and Process Industries, SINTEF, Trondheim, Norway, October 12–14, 2020

Keywords:
CFD, fluid dynamics, modelling

Cover illustration: Tapping of metal by Jan Erik Olsen

ISSN 2387-4295 (online)
ISBN 978-82-536-1684-1 (pdf)

SINTEF Academic Press
Address: Børrestuveien 3
PO Box 124 Blindern
N-0314 OSLO
Tel: +47 40 00 51 00

www.sintef.no/community
www.sintefbok.no

SINTEF Proceedings
SINTEF Proceedings is a serial publication for peer-reviewed conference proceedings on a variety of scientific topics.
The processes of peer-reviewing of papers published in SINTEF Proceedings are administered by the conference organizers and proceedings editors. Detailed procedures will vary according to custom and practice in each scientific community.

CFD APPROACH TO SIMULATE TWO PHASE FLOW INLINE-SEPARATOR COUPLING IBM, LES, LAGRANGIAN TRACKING AND VOF METHODS

Hanane ATMANI[1*], **Rémi ZAMANSKY**[1], **Eric CLIMENT**[1], **Dominique LEGENDRE**[1]
[1]Institut de Mécanique des Fluides de Toulouse (IMFT), Université de Toulouse, CNRS.
2 Allée du Professeur Camille Soula, Toulouse, 31400, France.

[*] E-mail: hanane.atmani@imft.fr

ABSTRACT

Inline fluid separation using a swirl element is a recent technology for oil/gas processing. Centrifugal forces up to 100 times the gravitational acceleration separate the phases, leaving the heavy phase close to the wall and the light one in the center. The current study is part of a Europeen project TOMOCON aiming at developing CFD methods in the in-house code JADIM to simulate the two-phase flow separation in order to help the development of inline separation control. The objective is to propose a hybrid approach based on Navier Stokes solver that makes possible accurate simulations with coarse spatial resolution. First, Immersed Boundary Method (IBM) is used to simulate both the pipe and the complex geometry of the swirl element on a cartesian regular mesh. Turbulence is modeled by the classical dynamic Smagorinsky sub-grid model in Large Eddy Simulation (LES) with a special stochastic wall law coupled to the IBM allowing to avoid the need for a mesh refinement in the near wall region. A Lagrangian tracking (LT) method is used to solve the dispersed bubbly flow and it is coupled to the Volume of Fluid (VoF) approach once the coalescence takes place and the gas core is formed. The numerical strategy based on the coupling of these different methods is presented and we report some of the simulations used for the verification-validation of the numerical developments.

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

NOMENCLATURE

Greek Symbols
ρ Density, $[kg/m^3]$
μ Dynamic viscosity, $[kg/ms]$
ν Kinematic viscosity, $[m^2/s]$
α_{IBM} Solid volume fraction , $[-]$
Δt Time step, $[s]$.

Latin Symbols
p Pressure, $[Pa]$.
u_i Velocity, $[m/s]$.
u_b Bulk velocity, $[m/s]$.
v_b Bubble velocity, $[m/s]$.
R pipe radius, $[m]$.
r_b Bubble radius, $[m]$.
V_b Bubble volume, $[m^3]$.
C_M Added mass coefficient, $[-]$.
C_D Drag coefficient, $[-]$.
C_L Lift coefficient, $[-]$.
C Phase fraction, $[-]$.

Sub/superscripts
f Continuous phase.
b Dispersed phase: bubble.

INTRODUCTION

Two-phase flow separation is a common process in many industrial applications. In the oil/gas extraction, for instance, the separation can be done using gravity or a centrifugal force. Based on the latter concept, the inline separator (see figure 1), through the swirl element inserted inside the pipe, pushes the heavy phase toward the wall, leaving the light phase in the center of the separator. This type of swirling flow, having complex features, is interesting to investigate especially when experimental studies become constrained to cost and time. CFD methods allow to understand more in details the flow characteristics particulary those influencing on the separation efficiency. Neverthless, considering all the range of flow and interface length scales present in the separator, new CFD approaches need to be developed.

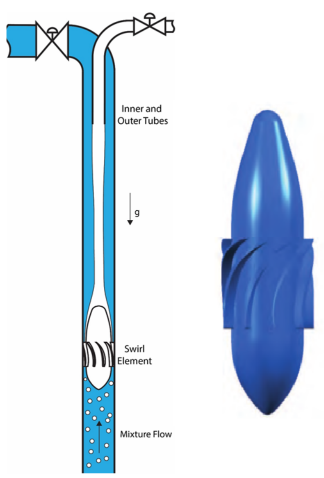


Figure 1: Sketch of the inline separator (left), swirl element (right)

Indeed, the interface scales are typically ranging from 1 m the length of the device (pipe, swirl element) to 10^{-4} m the size of the smallest bubbles and drops. The flow is highly turbulent ($Re = 10^4 - 10^6$), and the swirl element geometry is complex. The numerical strategy proposed here combines the Immersed Boundary Method (IBM) for the complex geometry, the Large Eddy Simulation (LES) for the turbulent flow, the Lagrangian Tracking (LT) for the dispersed phase and the Volume of Fluid (VoF) for the interface of the core formed in the separator wake. To simulate complex geometries, many methods exist for solid/fluid interaction. Peskin (Peskin, 1977) was the first to introduce Immersed Boundary Method (IBM) to study flow patterns around heart valves. This method is based on adding the force applied by the solid on the fluid to Navier Stokes equation and to locate the surface of the solid, Lagrangian markers are defined on the Eulerian grids. Bigot (Bigot *et al.*, 2014) replaced the use of those Lagrangian points by a solid volume fraction which is 0 is the fluid, 1 in the solid and between 1 and 0 through the interface. This enables to reduce the cost of the computation and provides a smooth transition across the fluid and the solid.

Considering the large Reynolds number of the flow in the process, the flow is turbulent. Therefore, performing Direct Numerical Simulation (DNS) becomes computationally impossible as the Reynolds number increases. In fact, to solve all time and 3D spatial scales, DNS requires a small time step and a number of the cell $N = O(Re^{9/4})$. For turbulence modeling we consider the LES method which models only scales below the filter size (the grid size). One of the interesting models in LES and which is suitable to confined flows is the mixed dynamic Smagorinsky model (Calmet and Magnaudet, 1997). This approach has been proved to handle very well the turbulent viscosity next to the wall through the calculation of a local Smagorinsky coefficient instead of using a constant value in the whole domain. However, a resolved LES always demands that 4 to 5 cells should be located in the viscous sub-layer. Therefore, it requires a significant mesh refinement close to the wall because the thickness of the viscous sub-layer decreases with the increase of the Reynolds number. To avoid this constraint, wall models are oftenly used and special treatments of the wall are done when the latter is simulated using IBM. Simplifying the Navier Stokes equations to have the thin boundary layer equation, Ma (Ma *et al.*, 2019) solved this equation on an embedded mesh to get the local wall shear stress at the IB cells and then corrected the SGS viscosity to take into account the IB forcing. In our work, a stochastic model for the velocity imposed inside the IBM boundary layer is used (Atmani *et al.*, 2020). This model makes possible an accurate resolution of both the mean velocity and the rms fluctuations on a coarse mesh in the bulk of a high Reynolds turbulent pipe flow.

Two phases are present in the separator and they are organized as dispersed before the separator and stratified flow in the separator wake. Each phase has to be treated using a specific approach: a LT method for the dispersed phase and a VoF method for the gas core in the separator wake. To take advantage from both methods, hybrid approaches have been recently proposed to make possible a dynamic switch. This kind of model is interesting when the bubbles/droplets are accumulated to form a core or when the VoF core is dispersed into small bubbles/droplets, as observed in two phase flow separation.

MODEL DESCRIPTION

The numerical approach is based on the IMFT-in-house CFD code JADIM. It solves the Navier-Stokes equations for incompressible, unsteady, three dimensional flows (Calmet and Magnaudet, 1997) (Legendre and Magnaudet, 1998):

$$\frac{\partial u_i}{\partial x_i} = 0 \tag{1}$$

$$\frac{\partial u_i}{\partial t} + \frac{\partial u_i u_j}{\partial x_j} = -\frac{1}{\rho}\frac{\partial p}{\partial x_i} + \nu \frac{\partial^2 u_i}{\partial x_j \partial x_j} + f_i \tag{2}$$

with f_i is the sum of any existing volumetric force applied on the fluid.

The diffusive term is solved using a semi implicit Crank Nicolson scheme, while the other terms are treated explicitly by Runge Kutta 3 and the projection method is used to satisfy the incompressibility condition 1.

Large Eddy Simulation

Using the mixed dynamic Smagorinsky model (Calmet and Magnaudet, 1997), the filtered Navier Stokes equations are:

$$\frac{\partial \overline{u}_i}{\partial x_i} = 0 \tag{3}$$

$$\frac{\partial \overline{u}_i}{\partial t} + \frac{\partial \overline{u}_i \overline{u}_j}{\partial x_j} = -\frac{1}{\rho}\frac{\partial \overline{p}}{\partial x_i} + \nu \frac{\partial^2 \overline{u}_i}{\partial x_j \partial x_j} - \frac{\partial \tau_{ij}^{SGS}}{\partial x_j} + \overline{f}_i \tag{4}$$

where $\overline{u}_i$ with (i=1,2,3) is now the filtered velocity, $\overline{p}$ is the filtered pressure, $\overline{f}_i$ is the filtered body force.

$\tau_{ij}^{SGS} = \overline{u_i u_j} - \overline{u}_i \overline{u}_j$ is the sub-grid stress tensor (SGS). It is can be expressed as the sum the following terms:

$$L_{ij} = \overline{\overline{u}_i \overline{u}_j} - \overline{\overline{u}}_i \overline{\overline{u}}_j \tag{5}$$

$$C_{ij} = \overline{\overline{u}_i u'_j} + \overline{\overline{u}_j u'_i} - \overline{\overline{u}}_i \overline{u'}_j - \overline{\overline{u}}_j \overline{u'}_i \tag{6}$$

$$R_{ij} = \overline{u'_i u'_j} - \overline{u'}_i \overline{u'}_j \tag{7}$$

The Leonard term L_{ij} is calculted explicitly.

$$\tau_{ij}^{SGS} - \frac{1}{3}\tau_{kk}^{SGS}\delta_{ij} = -2\nu_T \overline{S}_{ij} + L_{ij} - \frac{1}{3}L_{kk}\delta_{ij} \tag{8}$$

where $\overline{S}_{ij}$ is the strain rate tensor calculated from the resolved velocity field and the turbulent viscosity ν_T is given by:

$$\nu_T = C\overline{\Delta}^2 (2\overline{S}_{ij}\overline{S}_{ij})^{\frac{1}{2}} \tag{9}$$

with $\overline{\Delta}$ is the filter length. C is a local parameter calculated at each time step. By re-filtering eq (4) using $\widetilde{\Delta}$, we define: $T_{ij} = \widetilde{\overline{u_i u_j}} - \widetilde{\overline{u}}_i \widetilde{\overline{u}}_j$. Similary as τ_{ij}^{SGS}, T_{ij} is also expressed in function of C as:

$$T_{ij} - \frac{1}{3}T_{kk}\delta_{ij} = -2C\widetilde{\overline{\Delta}}^2 |\widetilde{\overline{S}}|\widetilde{\overline{S}}_{ij} + L_{ij}^T - \frac{1}{3}L_{kk}^T\delta_{ij} \tag{10}$$

with $L_{ij}^T = \widetilde{\overline{\overline{u}_i \overline{u}_j}} - \widetilde{\overline{\overline{u}}}_i \widetilde{\overline{\overline{u}}}_j$. Both T_{ij} and $\widetilde{\tau^{SGS}}_{ij}$ cannot be calculated explicitly however the difference $l_{ij} = T_{ij} - \widetilde{\tau^{SGS}}_{ij} = \widetilde{\overline{u}_i \overline{u}_j} - \widetilde{\overline{u}}_i \widetilde{\overline{u}}_j$ can be and allows to find the local coefficient C:

$$l_{ij} - \frac{1}{3}l_{kk}\delta_{ij} = -2C(\widetilde{\overline{\Delta}}^2 |\widetilde{\overline{S}}|\widetilde{\overline{S}}_{ij} - \overline{\Delta}^2 \widetilde{|\overline{S}|\overline{S}_{ij}}) - \widetilde{\overline{\overline{u}}}_i \widetilde{\overline{\overline{u}}}_j + \widetilde{\overline{\overline{u}}_i \overline{\overline{u}}_j} + \frac{1}{3}(\widetilde{\overline{\overline{u}}}_k \widetilde{\overline{\overline{u}}}_k - \widetilde{\overline{\overline{u}}_k \overline{\overline{u}}_k})\delta_{ij} \tag{11}$$

C is then:

$$C = -\frac{(l_{ij} - h_{ij})M_{ij}}{2M_{ij}M_{ij}} \tag{12}$$

with $M_{ij} = \widetilde{\overline{\Delta}}^2 |\widetilde{\overline{S}}|\widetilde{\overline{S}}_{ij} - \overline{\Delta}^2 \widetilde{|\overline{S}|\overline{S}_{ij}}$ and $h_{ij} = \widetilde{\overline{\overline{u}}_i \overline{\overline{u}}_j} - \widetilde{\overline{\overline{u}}}_i \widetilde{\overline{\overline{u}}}_j$

Immersed Boundary Method

To simulate the fluid/solid interaction, we use Immersed Boundary Method (IBM) by adding to Navier-Stokes equation a volumetric force $\overline{f}_{IBM}$ (Bigot *et al.*, 2014):

$$\overline{f}^k_{IBM,i} = \alpha_{IBM} \frac{\overline{vs}^{k-1}_i - \widehat{u}^k_i}{\Delta t} \tag{13}$$

with vs_i is the local velocity imposed to the solid, $\widehat{u}_i$ is the predictor fluid velocity without considering the solid. α_{IBM} is defined by a mathematical expression allowing to describe the geometry (Yuki and Takeuchi, 2007):

$$\alpha_{IBM}(x) = \frac{1}{2}[1 - \tanh(\frac{(\|\mathbf{x} - \mathbf{x}_p\| - R)}{\sqrt{2}\lambda\eta\Delta})] \tag{14}$$

with $\lambda = \mid n_x \mid + \mid n_y \mid + \mid n_z \mid$ is calculated using the components of $\mathbf{n}$ the normal outward unit vector at the surface. $\eta = 0.065(1 - \lambda^2) + 0.39$ is a parameter controlling the thickness of the transition region. A new stochastic wall model has been developed to enhance the forcing term of IBM through a correction of the solid velocity based on a log law inside the transition from solid to fluid taking into account the spatio-temporal correlation of the wall shear stress for the simulation of turbulent flows (Atmani *et al.*, 2020).

Lagrangian tracking

Lagrangian tracking consists in solving the trajectory equation for each bubble/droplet taking into account the buoyancy force, the drag force, the lift force, the added mass and Tchen forces (Chouippe *et al.*, 2014). Each bubble/droplet trajectory is obtained by solving:

$$\frac{d\mathbf{x}_b}{dt} = \mathbf{v}_b \tag{15}$$

where the bubble/droplet velocity is obtained by integrating the force balance:

$$\begin{aligned}(\rho_b + C_M\rho_f)V_b\frac{dv_{p,i}}{dt} &= (\rho_b - \rho_f)V_b g_i + \\ C_D\rho_f\frac{\pi r_p^2}{2}\|u - v_b\|(u_i - v_{b,i}) &+ C_L\rho_f V_b(u_i - v_{b,i}) \wedge \Omega + \\ &\rho_f V_b(1 + C_M)\frac{Du_i}{Dt}\end{aligned} \tag{16}$$

When considering the LES approach, the fluid velocity $u_{\mathbf{x}_b}$ and acceleration $Du/Dt|_{\mathbf{x}_b}$ at the bubble location $\mathbf{x} = \mathbf{x}_b$ are obtained by a second order interpolation of the filtered velocity $\overline{u}$. A two way coupling approach is also considered to take into account the effect of the bubbles/droplets on the fluid. Moreover, the bubbles/droplets can be in interaction with the solid wall described using the IBM method and a collision model has been introduced to model the rebound on the wall with a restitution coefficient r (is 1 for full restitution):

$$v^{new}_{b,i} = v_{b,i} - (1 + r)(\mathbf{n}.\mathbf{v}_b)n_i \tag{17}$$

where $\mathbf{n}$ is the solid surface normal oriented to the fluid.

Volume of Fluid

The Volume of Fluid (VoF) approach considered here is an interface capturing method without an interface reconstruction allowing to simulate the break-up and the coalescence. It is based on the VoF volume fraction C which equals 1 in one phase and 0 in the other. The value of C varies between 0 and 1 across the interface and is governed by the transport equation solved using the FCT (Flux Corrected Transport scheme) (Zalesak, 1979) (Bonometti and Magnaudet, 2007):

$$\frac{\partial C}{\partial t} + u_i.\frac{\partial C}{\partial x_i} = 0 \tag{18}$$

The capillarity force is added to Navier Stokes equation and is solved using the classical CSF (Continuum Surface Force) model intoduced by Brackbill(Brackbill *et al.*, 1992):

$$F_\sigma = \sigma\nabla.(\frac{\nabla C}{\|\nabla C\|})\nabla C \tag{19}$$

The hybrid LT/VoF model is activated when the coalescence of the bubbles occurs i.e. after the swirl element. The algorithm for switching from Lagrangian tracking to VoF is decomposed as following:
1- Identify any bubble/droplet verifying the coalescence criteria i.e. inside a cell where $0 < C < 1$.
2- The detected bubbles/droplets are removed from the Lagrangian solver and the phase fraction C is updated by adding the volume of those bubbles/droplets.

RESULTS

The objective of this paper is to present the preliminary results of our approach consisting in coupling the methods IBM, LES, LT and VoF and to report some of the simulations used for the verification-validation of the numerical developments. The numerical domain used for the system considered in this work is of size $L_x \times L_y \times L_z = 0.9m \times 0.104m \times 0.104m$ on a regular cartesian mesh made of $N_x \times N_y \times N_z = 800 \times 92 \times 92$ cells in the x ,y and z directions, respectively. The pipe diameter is $D = 2R$ and its axis is along the x-direction. First, we have to describe the complex geometry of the separator system using the IBM as reported in Figure 2. The flow field has been simulated for

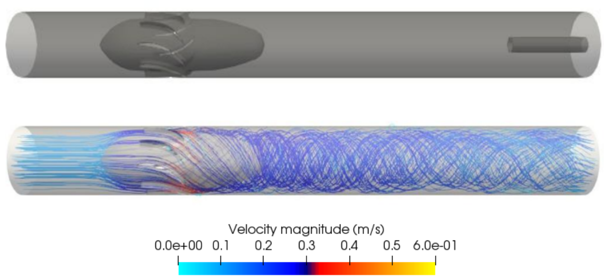


Figure 2: (Top) Description of the pipe, the separator and the pick-up tube using IBM. The contour $\alpha_{IBM} = 0.5$ is shown on the figure. (Bottom) Streamlines of single phase flow in the separator for $Re = 4500$.

different Reynolds number defined using the pipe diameter $D = 2R$ and the pipe mean velocity u_b. Figure 2 represents the streamlines for $Re = \rho D u_b/\mu = 4500$, u_b being the bulk velocity. The velocity field is clearly swirled by the separator and a vortex is formed in the separator wake. The velocity reaches its maximum next to the blades of the swirl element. To characterize the strength of the swirling flow, we introduce the dimensionless swirl number S defined as the ratio of the axial flux of the angular momentum to the axial flux of the axial bulk momentum:

$$S(x) = \frac{\int \rho r u_x u_\theta dA}{R\rho u_b^2 A} \tag{20}$$

where u_x is the axial velocity, u_θ is the angular velocity and A is the pipe section. Figure 3 presents the variation of the swirl number S along the pipe. S is maximum at the end of the swirl element and then it decreases following the relation obtained experimentally (Dirkzwager, 1996):

$$S(x) = S_0 \exp(-C_{ds}(x - x_0)/D) \tag{21}$$

where S_0 is the swirl number at the reference point x_0 right after the swirl element, D is the pipe diameter and C_{ds} is a swirl decay coefficient depending on Re, the roughness of the pipe and the swirl intensity. Figure 4 presents the radial

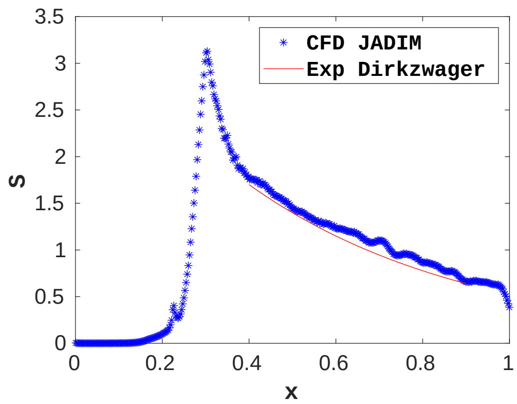


Figure 3: The swirl number S in function of the position x, fitting in red line based on Dirkzwager results where: $x_0 = 0.4$, $S_0 = 1.32$, $C_{ds} = 0.21$.

distribution of the dimensionless time-averaged axial and azimuthal velocities as well as pressure downstream the swirl element for a distance of D/2, 2D, 3D. The centrifugal force on the fluid, acting towards the wall, is the reason why the pressure drops in the center. In addition we can easily see that the pressure increases with the increase of the azimuthal velocity. The axial velocity profile shows the occurence of a reverse flow behind the swirl element, a phenomenon which characterizes swirling flows in general.

Now, a set of 10^5 bubbles of radius 1mm are injected at the inlet with the fluid velocity (see figure 5). The simulation shows that a relatively small group of bubbles is accumulated around the center of the pipe. The lift force induced by the swirling flow is expected to induce the migration of the bubbles to the pipe center. The magnitude of the lift force is directly proportional to the magnitude of the vortex generated in the separator wake. Its magnitude is expected to increase with increasing Reynolds number. Simulation of high Reynolds number flow has required the development of a specific wall modeling for LES simulations applied to the IBM wall zone ($0 < \alpha_{IBM} < 1$) (Atmani *et al.*, 2020). The proposed LES-IBM modeling allows an accurate simulation of the mean and rms velocities in the bulk of high Reynolds turbulent pipe flows with a coarse mesh (typically, a grid size of order 100 times the wall unit for $Re = 10^6$). Finally to make possible the global simulation of all the physical mechanisms involved in the separation process, the proposed coupling between the LT and VoF solvers has to be validated. For that purpose, the numerical set up is simplified and we model the velocity fields in the separator wake by considering a Taylor Couette flow generated by the pipe wall rotation. Bubbles are randomly injected. Figure 6 illustrates the bubbles accumulation towards the pipe center resulting in an air core formation.

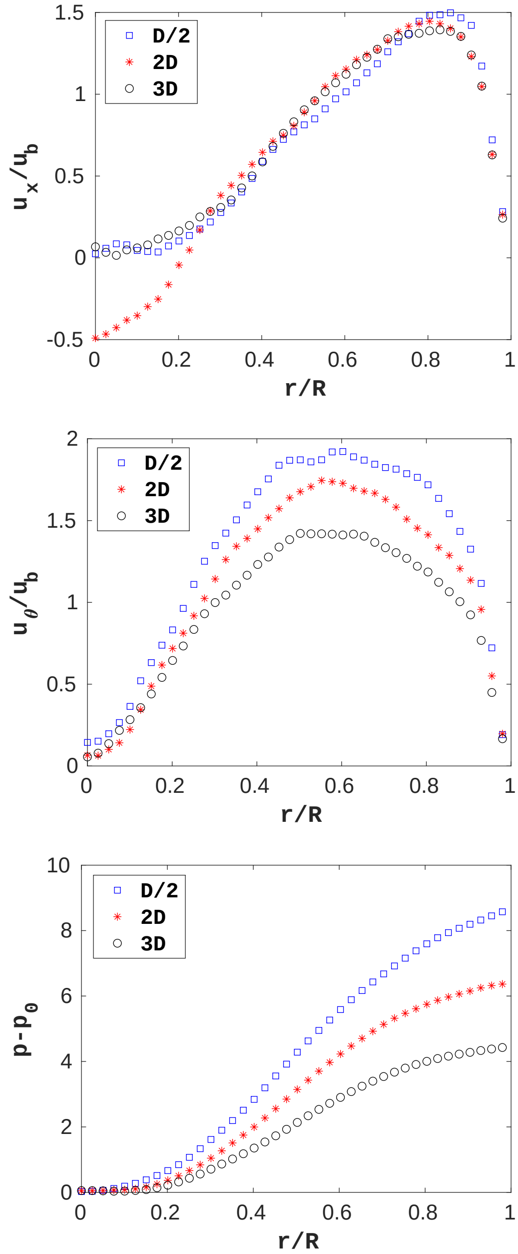


Figure 4: The radial distribution of the axial, the azimuthal velocities and pressure.

Figure 5: Simulation of two phase flow.

CONCLUSION

We have presented the numerical strategy developed to make possible accurate simulations of the inline separation with limited CPU cost. The complexity of the geometry and the large characteristic length scales involved in the process for both the turbulent flow and the interface topology (from dispersed to separated) make the numerical simulation challenging. The numerical strategy proposed here combines the Immersed Boundary Method (IBM) for the separator geometry, the Large Eddy Simulation (LES) for the turbulent flow, the Lagrangian Tracking (LT) for the dispersed phase and the Volume of Fluid (VoF) for the interface of the core formed in

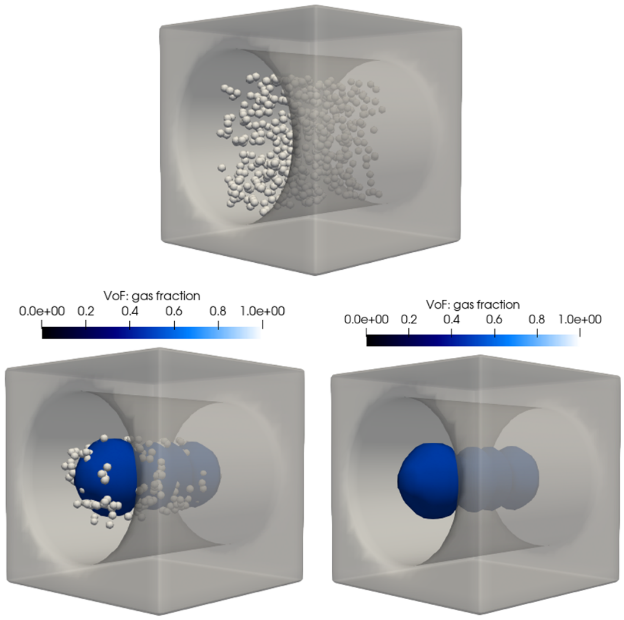


Figure 6: (top): Initial condition for the bubbles position. (bottom left) bubbles are accumulating at the center and are generating the gas core. (bottom right) All bubbles have been captured inside the air core.

the separator wake. The objective of this paper was to present the different steps of validation with a particular attention to the coupling between the methods. Some validations still need to be carried out in particular for the core formation in the separator wake for large Reynolds numbers. Numerical results of the entire process will then be compared to the experimental data produced by our partners TU Delft, TUL and HZDR in the TOMOCON H2020 project.

ACKNOWLEDGMENTS

This work is done within the TOMOCON project which has received funding from the European Union's Horizon 2020 research and innovation programme under the Marie Sklodowska Curie grant agreement No 764902. Numerical simulations were performed using high-performance computing resources from CALMIP Center of the University of Toulouse.

REFERENCES

ATMANI, H., ZAMANSKY, R., PEDRONO, A., CLIMENT, E. and LEGENDRE, D. (2020). "Stochastic wall model for turbulent pipe flow using immersed boundary method and large eddy simulation". *Computers and fluids, (in progress)*.

BIGOT, B., BONOMETTI, T., THUAL, O. and LACAZE, L. (2014). "A simple immersed boundary method for solid fluid interaction in constant and stratified density flows". *J. Computers and fluids*.

BONOMETTI, T. and MAGNAUDET, J. (2007). "An interface capturing method for incompressible two-phase flows. val- idation and application to bubble dynamics". *International Journal of Multiphase Flow*.

BRACKBILL, J., KOTHE, D. and ZEMACH, C. (1992). "A continuum method for modeling surface tension." *J Comput Phys*.

CALMET, I. and MAGNAUDET, J. (1997). "Large eddy simulation of high schmidt number mass transfer in a turbulent channel flow". *Physics of Fluids*.

CHOUIPPE, A., CLIMENT, E., LEGENDRE, D. and GABILLET, C. (2014). "Numerical simulations of bubble dispersion in turbulent taylor couette flow". *Physics of Fluids*.

DIRKZWAGER (1996). *A new axial cyclone design for fluid separation*. Ph.D. thesis, Delft university of Technology.

LEGENDRE, D. and MAGNAUDET, J. (1998). "The lift force on a spherical bubble in a viscous linear shear flow." *J. Fluid Mech.*

MA, M., HUANG, W. and XU, C. (2019). "A dynamic wall model for large eddy simulation of turbulent flow over complex/ moving boundaries based on the immersed boundary method". *Physics of Fluids*.

PESKIN, C. (1977). "Numerical analysis of blood flow in the heart". *J. Comput. Phys*.

YUKI, Y. and TAKEUCHI, S. (2007). "Efficient immersed boundary method for strong interaction problem of arbitrary shape object with the self-induced flow". *Journal of Fluid Science and Technology*.

ZALESAK, S.T. (1979). "Fully multidimensional flux corrected transport algorithms for fluids". *Journal of computational physics*.