# Literature Review

Multiphase flow can be found in many operational environments in the Oil & Gas industry. From the flow through porous media in the reservoir, the flow of reservoir fluids through the productions wells, the handling of hydrocarbon products in the production facilities, and the transportation to the final customers, it is required a good understanding of the flow behavior. The complexity of the turbulent two-phase flow makes the theoretical solution extremely hard; thus, experimental approaches have been used to descript the phenomena and predict the flow condition. However, implementing numerous experiments consumes a lot of resources. For this reason, numerical studies have become the generalized approach in the research on multiphase flow.

Solving a Computational Multi-Fluid Dynamic (CMFD) problem involves many areas of the engineering. First, the researcher has to decide which multiphase flow model is suitable for his problem. In general, this model consists in a system of partial differential equations as a result to apply the conservation laws of physics. Then, the computational model used to approximate the solution must be chosen. This task is not straightforward if we consider that for the last 40 years many computational model have arisen and their performance (accuracy and convergence speed) is still a topic of discussion.

## Multiphase Flow Models

Solving the full Navier-Stokes equations with the presence of a deforming phase boundary is challenging. Different multiphase flow models have been proposed and successfully implemented to solve numerically the multiphase flow phenomena. In order of complexity, we can list then as Fractional Volume of Fluid (VoF), Level Set (LS), Front Tracking (FT), Lattice Bolztman (LB) Methods and the Filtered Navier-Stokes Equation approach.

First, the bubble tracking problem was successfully model using a Lagrangian description, however the results related with time averaged velocity field, turbulence intensity, turbulent viscosity and gas hold-up profiles suggested that the Euler-Lagrangian model is applicable only at lower gas-flow rates (Ashraf Ali and Pushpavanam 2011). The Volume of Fluid uses a Eulerian formulation for problems that involve deformed free boundaries. The interphase is captured by keeping track of the volume fraction of each computational cell in two- or three-dimensional meshed with respect to one of the fluid phases (Hirt and Nichols 1981). This approach has been applied, inter alia, in open channels problem (Hirt and Nichols 1981); multiphase flow in cylindrical vessel (Peng, et al. 2010); unsteady gas-liquid flows in a rectangular tank (Ashraf Ali and Pushpavanam 2011).

The main difficulty in using this method has been the maintenance of a sharp boundary between the different fluids and the computation of the surface tension (Tryggavason, et al. 2001). In order to address this problem, (Brackbill, Kothe and Zemach 1992) developed a technique to include the surface tension in the model and (Chen, et al. 1997) used subcells to improve the resolution of the interface. The Level Set method (LS) also arise as a possible solution.

In LS, the interface is considered to be a level surface of a function that is define over all space. LS consist in solving the topology equation in a conventional way, while introducing a subtle mean for localizing the interface on the grid. A smooth function is defined everywhere in the domain, referring to the shortest distance to the from. Negative values correspond to one of the fluids and positive values to the other. The exact location of the interface corresponds to the zero level of **.** (Osher and Sethian 1988)**.** Nevertheless, the method suffers from a lack of mass conservation (Szewc, Pozorski and Minier 2013). Two-dimensional and axisymmetrical simulations of rising bubbles using LS were presented by (Sussman, Smereka and Osher, A Level Set Approach for Computing Solutions to Incompressible Two-Phase Flow 1994) and (Sussman and Smereka, Axysymmetric free boundary problems 1997)

On the other hand, the Front Tracking methods use a fixed grid and a separate front which marks the interface. The grid mesh is refined near the front to make a grid line to follow the interface (Glimm, et al. 2001). (Tryggavason, et al. 2001) implemented successfully a hybrid model between a front capturing and a front-tracking technique for solving a homogeneous bubbly flow, atomization, flows with variable surface tension, solidification, and boiling problems.

Currently, the computational models are using a filtered Navier-Stokes equations model. The first contributions introducing the derivation of the filtered multi-fluid flow equations under isothermal flow conditions were in (Lakehal, Smith and Milelli 2002) and shortly after in (Sirignano 2005), who generalized the strategy to further cope with reactive flows. Both contributions propose an extension of filtering for turbulent flows to combine interfacial and turbulence scales into one unified filter (Lakehal 2018).

## Computational Models

As the capabilities of the computers got improved, researchers were able to implement new techniques to solve turbulent flow problems. Then, many of those techniques were extrapolate to the multiphase condition. Galerking Method emerged as an intent to overcome the computational efficiency of the traditional finite different methods. This was accomplished using a base of eigenfunctions instead of the fixed base used in the traditional grid box. (Davies and Stephens 1983) showed that the rate of convergence of the Galerkin method with a basis set of Chebyshev polynomials is far superior to that found using uniform grid box spacing applied to an oceanographic flow problem.

Different researchers have implemented Direct Numerical Simulation (DNS) in multiphase flow problems in cases of homogenous turbulence. (Elghobashi and Truesdell 1992) addressed the phenomena of particle flow, (Boivin, Simonin and Squires 1998) presented a solution for droplet flow and (Druzhinin and Elghobashi 2001) studied the case for bubble-laden flow. However, whether the droplets are average together as a group and represented by an average droplet or they are represented individually as point sources, there is a loss of resolution on the scale of the droplet and its surrounding film. So, the DNS calculations require modelling of the behavior at the droplet scales and implicitly are restricted to situations where the smallest eddy scales are at least an order of magnitude larger than the droplet scales (Sirignano 2005).

In the Reynolds-Averaged Navier Stoked (RANS) modelling framework, the instantaneous fluid velocities are decomposed into a mean velocity  , and a fluctuating component . The mean part is then obtained directly from the time-averaged Eulerian solution (deterministic), whereas the fluctuating part must be obtained separately through stochastic modelling (Njobuenwu, Fairweather and Yao 2013). This approach was used by (Njobuenwu, Fairweather and Yao 2013) to solve a particle-laden flow problem in a duct with a 90° bend. The alternative to RANS models are models that resolve at least a portion of the turbulence for at least a portion of the flow domain. Such models are generally termed ‘Scale-Resolving’. The most known one is Large Eddy Simulation model.

The weaknesses of phase averaging to predict various (sometimes rather simple) types topologies, e.g. stratify slug flow, and also the limited predictive performance in the multiphase flow context of statistical turbulence modeling, motivated the transition toward scale-resolving turbulence simulation (SRS) (Lakehal 2018). In the study presented by (Garg, et al. 2007), the DNS and LES algorithms where tested solving a dispersed two-phase flow and using a Lagrangian-Eulerian multiphase flow model and the authors concluded that, in order to obtain accurate estimation of the momentum transfer term, is required using higher number of particles per cell.

LES has been the first SRS method and is under development for almost five decades, however has not replace completely RANS model. For wall boundary layers, the turbulence length scale becomes very small relative to the boundary layer thickness near the wall (increasingly so with increased Re number). This poses severe limitations for LESS (Menter, Schütze and Kurbatskii 2011)

Detached Eddy Simulation (DES) models switch explicitly between RANS and LES model formulation based on the local grid spacing and turbulent length scale. The original intent of DES was to be run in RANS mode for attached boundary layers and to switch to LES mode in large separated (detached) flow regions. DES models allow a local reduction in eddy-viscosity by grid refinement in the transition region between RANS and LES, which in turns can help in the formation of unsteady content (Gritskevich, et al. 2012).

Another variations of the LES techniques are Dispersed-Flow LES (LESS) and Interfacial-Flow LES (LEIS). LESS has been employed under the two-fluid and mixture model variants essentially for turbulent bubbly flow. The derivation of the LESS equations can be found in (Lakehal, Smith and Milelli 2002) and (Sirignano 2005); the latter considered heat transfer and chemical reaction. EIS has been applied to turbulent gas-liquid flows involving large-scale sheared interphases, with problem ranging from spilling wave flows to steam injection in water pool. Also, the LEIS concept has been updated to cope with mass transfer of phase-change induced by heat transfer problems under turbulent flow conditions. (Lakehal 2018).

LEIS alone would not capture the individual gas bubbles in a slug flow because of insufficient grid resolution. All-Regime Multi-Fluid model (ARM) unifies the approaches LESS and LEIS. The idea is to predict large resolved interfaces together with subscale dispersed entities that may be generated from the sheared interface itself.

# References

Amaya-Bower, Luz, and Taehun Lee. "Single bubble rising dynamics for moderate Reynolds number using Lattice Boltzmann Method." *Computers & Fluids. Vol. 39*, 2010: 1191-1207.

Ashraf Ali, B., and S. Pushpavanam. "Analysis of unsteady gas-liquid flows in a rectangular tank: Comparison of Euler-Eulerian and Euler-Lagrangian simulations." *International Journal of Multiphase Flow. Vol. 37*, 2011: 268-277.

Boivin, M., O. Simonin, and K.D. Squires. "Direct numerical simulation of turbulence modulation byb particles in isotropic turbulence." *Journal of Fluids Mechanics. Vol. 375*, 1998: 235-263.

Brackbill, J. U., D.B. Kothe, and C. Zemach. "A Continuum Method for Modeling Surface Tension." *Journal of Computational Physics. Vol. 100*, 1992: 335-354.

Chen, Shea, David B. Johnson, Raad Peter, and Dani Fadda. "The Surface Marker and Micro Cell Method." *International Journal for Numerical Methods in Fluids. Vol. 25*, 1997: 749-778.

Davies, A.M., and C.V. Stephens. "Comparion of the finite difference and Galerkin Methods as applied to the solution of the hydrodynamic equations." *Applied Mathematical Modelling. Volume 7*, 1983: 226-240.

Druzhinin, O.A., and S. Elghobashi. "Direct numerical simulation of a three-dimensional spatially developing droplet-laden mixing layer with two-way coupling." *Journal of Fluids Mechanics. Vol. 429*, 2001: 23-61.

Elghobashi, S., and G. Truesdell. "Direct Simulation of Particle Dispersion in Decaying Isotropic Turbulence." *Journal of Fluid Mechanics. Vol. 242*, 1992: 655-706.

Garg, R., C. Narayanan, D. Lakehal, and S. Subramanian. "Accurate numerical estimation of interphase momentum transfer in Lagrangian-Eulerian simulations of dispersed two-phase flows." *International Journal of Multiphase Flow. Vol. 33*, 2007: 1337-1364.

Glimm, J., J.W. Grove, X.L. Li, W. Oh, and D.H. Sharp. "A Critical Analysis of Rayleigh-Taylor Growth Rates." *Journal of Computational Physics. Vol. 169*, 2001: 652-677.

Gritskevich, Mikhail S., V. Garbaruk Andrey, Jochen Schütze, and Florian R. Menter. "Development of DDES and IDDES Formulations for the k-ω Shear Stress Transport Model." *Flow, Turbulence and Combustion. Vol. 88*, 2012: 431-449.

Hirt, C. W., and B. D. Nichols. "Volume of fluid (VOF) method for the dynamics of free boundaries." *Journal of Computational Physics*, 1981: 201-225.

Lakehal, Djamel. "Status and future developments of Large-Eddy Simulation of turbulent multi-fluid flow (LEIS and LESS)." *International Journal of Multiphase Flow. Vol. 104*, 2018: 322-337.

Lakehal, Djamel, Brian L. Smith, and Massimo Milelli. "Large-Eddy simulation of bubbly turbulent shear flows." *Journal of Turbulence, 3*, 2002: DOI: 10.1088/1468-5248/3/1/025.

Menter, F.R., J. Schütze, and K.A. Kurbatskii. "Scale-Resolving Simulation Techniques in Industrial CFD." *6th AIAA Theoretical Fluid Mechanics Conference.* Honolulu, Hawaii: American Institute of Aeronautics and Astronautics, 2011.

Morton, K.W., and David Mayers. *Numerical Solution of Partial Differential Equations. An Introduction. 2nd Edition.* Cambridge: Cambridge University Press, 2005.

Njobuenwu, D.O., M. Fairweather, and J. Yao. "Coupled RANS-LPT modelling of dilute, particle-laden flow in a duct with a 90° bend." *International Journal of Multiphase Flow. Vol. 50*, 2013: 71-88.

OpenFD Ltd. *OpenFOAM. The open source CFD toolbox.* 2004. https://www.openfoam.com/.

Osher, Stanley, and James A Sethian. "Fronts propagating with curvature-dependent speed: Algorithms based on Hamilton-Jacobi formulations. Vol. 79." *Journal of Computational Physis*, 1988: 12-49.

Peng, Zhengbiao, Zhulin Yuan, Tie Li, Jie Cai, Xuan Wu, and Fengxian Fan. "Research on multiphase flows in Thermo-Energy Engineering Institute of Southeast University." *International Journal of Multiphase Flow*, 2010: 119-126.

Sirignano, William A. "Volume averaging for the analysis of turbulent spray flows." *International Journal of Multiphase Flow. Vol. 31*, 2005: 675-705.

Stanford University. *SU2 The Open-Source CFD Code.* n.d. https://su2code.github.io/.

Sussman, Mark, and Peter Smereka. "Axysymmetric free boundary problems." *Journal of Fluid Mechanics*, 1997: 269-294.

Sussman, Mark, Peter Smereka, and Stanley Osher. "A Level Set Approach for Computing Solutions to Incompressible Two-Phase Flow." *Journal of Computational Physics. Vol. 114*, 1994: 146-159.

Szewc, K., J. Pozorski, and J.-P. Minier. "Simulation of single bubbles rising through viscous liquids using Smoothed Particle Hydrodynamics." *International Journal of Multiphase Flow. Vol. 50*, 2013: 98-105.

Tryggavason, G., et al. " A Front-Tracking Method for the Computations of Multiphase Flow." *Journal of Computational Physics. Volume 169*, 2001: 708-759.