**Student: Victor Pugliese**

**R#: 11492336**

**Computational Multi-Fluid Dynamics (CMFD)**

Content

[1 Literature Review 2](#_Toc524641264)

[1.1 Multiphase Flow Models 2](#_Toc524641265)

[1.1.1 Volume of Fluid Method 2](#_Toc524641266)

[1.1.2 Level Set Method (LS) 2](#_Toc524641267)

[1.1.3 Front Tracking 2](#_Toc524641268)

[1.1.4 Lattice Boltzman Methods 2](#_Toc524641269)

[1.1.5 The filtered Navier-Stokes equations 2](#_Toc524641270)

[1.2 Computational Models 2](#_Toc524641271)

[1.2.1 Direct Numerical Simulation (DNS) 2](#_Toc524641272)

[1.2.2 Reynolds-Averaged Navier Stokes (RANS) 2](#_Toc524641273)

[1.2.3 Large Eddy Simulation (LES) 2](#_Toc524641274)

[1.2.4 All-Regime Multi-fluid (ARM) model 3](#_Toc524641275)

[2 Computational tool 3](#_Toc524641276)

[2.1 Selection 4](#_Toc524641277)

[2.2 Numerical solution of Partial Differential Equations 4](#_Toc524641278)

[2.2.1 Parabolic Equation 2D 4](#_Toc524641279)

[2.2.2 Hyperbolic Equation 4](#_Toc524641280)

[2.2.3 Elliptic Equation (Diffusion equation) 4](#_Toc524641281)

[3 References 4](#_Toc524641282)

# Literature Review

## Multiphase Flow Models

### Volume of Fluid Method

The interphase is captured by keeping track of the volume fraction of each computational cell in the grid with respect to one of the fluid phases. (Hirt and Nichols 1981)

### Level Set Method (LS)

The interface is considered to be a level surface of a function that is define over all space (Osher and Sethian 1988). 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 **.**

### Front Tracking

### Lattice Boltzman Methods

### The filtered Navier-Stokes equations

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

### Direct Numerical Simulation (DNS)

Turbulence-scale resolving method (All-scales)

### Reynolds-Averaged Navier Stokes (RANS)

Statistical time averaging.

### Large Eddy Simulation (LES)

Larger scale than DNS. 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. (Lakehal 2018)

#### Very-Large Eddy Simulation

#### Detached Eddy Simulation

#### Dispersed-Flow LES (LESS)

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.

#### Interfacial-Flow LES (LEIS)

LEIS 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).

### All-Regime Multi-fluid (ARM) model

LEIS alone would not capture the individual gas bubbles in a slug flow because of insufficient grid resolution. 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.

## Questions

# Computational tool

Computational Fluid Dynamics is an integrating area where the disciplines of fluid mechanics and mathematics use the computer science developments for solving complex problems. Currently, there is a wide variety of CFD software packages (and continuously increasing the capability and performances), and sometimes is difficult to identify which one is suitable for the specific application. CFD software could be classified using the following categories:

***Open Source:*** Open-source software permits users to study, change, and improve the software. It remains free of cost and a large amount of developers help to keep the packages update. *OpenFOAM* and *SU2* are possibly the most known software used by researchers and developers around the world. “*SU2* is an open-source collection of software tools written in C++ and Python for the analysis of partial differential equations (PDEs) and PDE-constrained optimization problems on unstructured meshes with state-of-the-art numerical methods” (Stanford University n.d.). “*OpenFOAM* has an extensive range of features to solve anything from complex fluid flows involving chemical reactions, turbulence and heat transfer, to acoustics, solid mechanics and electromagnetics (OpenFD Ltd. 2004)

Open Source wrappers:

* Visual-CFD
* HELYX
* SimFLow

CAD integrated:

* Solidworks
* Autodesk Inventor

Specialty:

* Converge
* AVL Fire
* FloTHERM

Comprehensive Packages:

* Fluent
* Star-CCM+
* COMSOL

## Selection

<https://www.cfd-online.com/Forums/>

## Numerical solution of Partial Differential Equations

(Morton and Mayers 2005)

### Parabolic Equation 2D

#### Euler Explicit Scheme

#### Euler Implicit Scheme

#### Crank-Nicolson Method

### Hyperbolic Equation

#### Upwind Scheme

#### Lax-Wendroff Scheme

#### Finite Volume Scheme

#### Leap-frog Scheme

### Elliptic Equation (Diffusion equation)

#### Central Difference Scheme

# References

Hirt, C. W., y 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, y Massimo Milelli. «Large-Eddy simulation of bubbly turbulent shear flows.» *Journal of Turbulence, 3*, 2002: DOI: 10.1088/1468-5248/3/1/025.

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

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

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

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.* s.f. https://su2code.github.io/.