

# MPflow: Research-based multiphase CFD for enterprise

Advanced Multiphase Flow Analysis

Next-generation CFD for Multiphase Flows



## Services and product

- A research-based finite volume software designed to allow one to simulate complex flows efficiently allowing for various options for different dedicated multiphase flow solvers
- Taylor-made CFD solutions for multiple-fluids problems
- Based on OpenFOAM, validated, documented codes used for various problems with complex physics

## MPflow

- Simulation of two or more immiscible fluids with or without phase change. Phase-change models for boiling, evaporation, condensation, sublimation.
- Algorithms suitable for different types of mesh. State of the art interface capturing methods (level set, volume of fluid methods). Novel methods for mixtures of different fluids.
- Specifically designed models for cryogenic flows and LNG simulations. Dedicated solvers for combustion for non-premixed flames and explosions simulations.

## Capabilities

Particle/Dispersed flows	Free-surface/Internal flows
Break-Up And Coalescence	Improved VOF
Atomisation	Conservative Level Set
Evaporation	Coupled Level Set/VOF
VOF-Particle Tracking	Multi-fluids cavitation
Spray modelling	Flashing
Combusting Particles	HRM
Euler-Euler with cavitation	Surface Tension
Drag and Lift	Phase Change
ELSA model	Heat and Mass Transfer

- Coupled volume of fluid/level set for interface tracking of two immiscible fluids
- Three-phase mixtures with fluid/fluid/solid particle flows
- Pipe flows and atomisation for subcooled and superheated jets
- Dedicated solvers for simulating hazardous liquids and gases and accidental releases (LNG, CO<sub>2</sub>, ...)

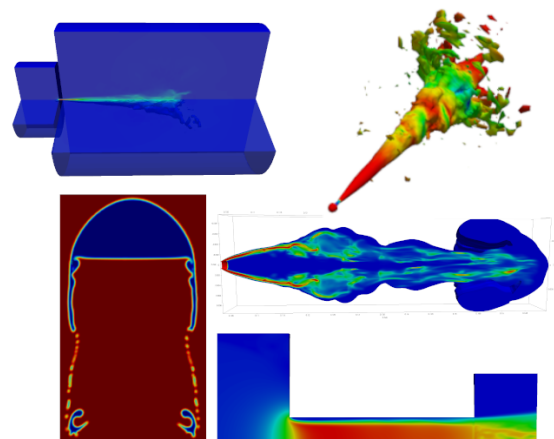


Figure 1: 3D CFD simulations of various problems

## User experience

- Solvers can be used for advanced calculations of mixtures, fundamental problems, large-scale applications and hazards and safety analysis
- All common OpenFOAM utilities available within different platforms for Linux, Mac OS X and Windows.
- For more details and for free testing visit <http://www.multifluidx.com>