

# BENCH MARK

## January 2018 issue ...

- Near-Future Directions of Multiscale & Multiphysics Modeling & Simulation
- Personalized Healthcare in the Age of Simulation
- Co-simulation - Art or Science?
- Large Deformation of Rubber Sleeves in Valves Using Strongly Coupled Fluid-Structure Interaction
- Multiphysics Analysis of Lithium Ion Battery
- What is The Functional Mock-up Interface?
- Aero-Vibro-Acoustics for Wind Noise Applications
- An Introduction to Multiphase Flow Modelling
- Why do I need Multiscale Modeling?
- Multiphysics of Sloshing in the Nuclear Industry
- Benchmarking Accuracy and Parallel Performance of an Open Source Framework for Fluid Structure Interaction Problems
- Building the Credibility of Engineering Simulation in Aerospace
- Co-Simulation - What is it?
- Excel for Engineers and other STEM Professionals

THE INTERNATIONAL MAGAZINE FOR ENGINEERING DESIGNERS & ANALYSTS FROM **NAFEMS**



Bringing it all Together  
Multiphysics, Multiscale & Co-Simulation



# An Introduction to Multiphase Flow Modelling

Nicolas Tonello, Renuda

**M**odelling the interaction of different fluid materials and phases in multiphase simulations is a prime example of coupling in CFD. There are different frameworks available for flows of this type and this article provides a general overview of the options available.

Generally, in CFD we take multiphase flow to involve the flow of more than one material in the computational domain of interest. The strict thermodynamic definition of a phase is a gas, liquid or solid but for engineering simulation, specifically CFD, we broaden it to mean multiple materials with different characteristics. They may be different phases, such as sand being carried along in water, or the same phase, such as oil and water. In some cases, different categories of a single material might even be considered as separate 'phases' for simulation, such as spray from the tops of ocean waves, in addition to the water and air phases or different sized solid particles, grouped according to their behaviour. The combinations are endless and theoretically there is no limit to the number of different 'phases' that are considered.



Generally, in CFD we take multiphase flow to involve the flow of more than one material in the computational domain of interest. The strict thermodynamic definition of a phase is a gas, liquid or solid but for engineering simulation, specifically CFD, we broaden it to mean multiple materials with different characteristics. They may be different phases, such as sand being carried along in water, or the same phase, such as oil and water. In some cases, different categories of a single material might even be considered as separate 'phases' for simulation, such as spray from the tops of ocean waves, in addition to the water and air phases or different sized solid particles, grouped according to their behaviour. The combinations are endless and theoretically there is no limit to the number of different 'phases' that are considered.

## Multiphase Flows

CFD analysis of single phase flow involves modelling only one material, for example water flowing in a pipe or air moving around an automobile in a wind tunnel. However, as soon as different materials are involved which may behave differently from each other, the specifics of each material must be taken into account – this is what multiphase flow modelling seeks to achieve. For example, add rain to the air around the automobile and multiphase flow modelling will be required if one wants to look at the impact of the air and the rain drops on the windows. To understand the interest and challenges in modelling multiphase flows numerically, it is interesting to first consider a few examples of such flows.

Multiphase flows are found everywhere, in nature and industry. In nature, a spectacular example which illustrates some of the multiphase flow modelling challenges can be found in volcanic eruptions, which involves many different materials, such as rocks, dust and gases all being vented together out of the crater.

It could be of interest to carry out CFD simulations of the eruption in order to predict the impact of dust on the local communities or the concentration of noxious gases. As they behave differently and may influence one another, a CFD simulation of such an event would have to consider all the different materials. In order to apply specific models and properties to each of them, for example the size and density of the dust or the rocks, one would then classify them as different phases within one multiphase flow calculation. The motion of the different phases would then be calculated as they move and interact with each other in time and space. The photograph in Figure 1 also exemplifies the variety of spatial scales and physical phenomena which can be encountered in multiphase flow, from the plume of debris and gases close to the volcano to the formation of clouds and their interaction with atmospheric patterns. To correctly capture the motion of all the phases, one would need to resolve the coupling between the phases on a range of physical and time scales, for example how the dust is transported by large scale atmospheric currents, but also how it might interact with turbulent eddies.

Other notable examples involve the air-water flows witnessed in oceans. Figure 2 shows strong interaction between the ocean and the surrounding air. The wind interacts with the water surface, helping to shape the waves and deforming their surface. At the crest of the waves, breakup is visible and sprays are formed, which are further picked up by the wind. In a simulation, two or three phases could be used to represent the different components, one for the ocean water, another one for the air and, possibly, a third one for the water droplets to represent spray.

In industry, multiphase flow phenomena may be an inherent and integral part of the systems' design. For example, in an internal combustion engine, fuel is atomised and sprayed in very fine droplets in the combustion chamber. This helps disperse the fuel over a



Figure 1: A Volcanic Eruption



Figure 2: Air-Water Interaction Taking Place on the Crest of a Wave



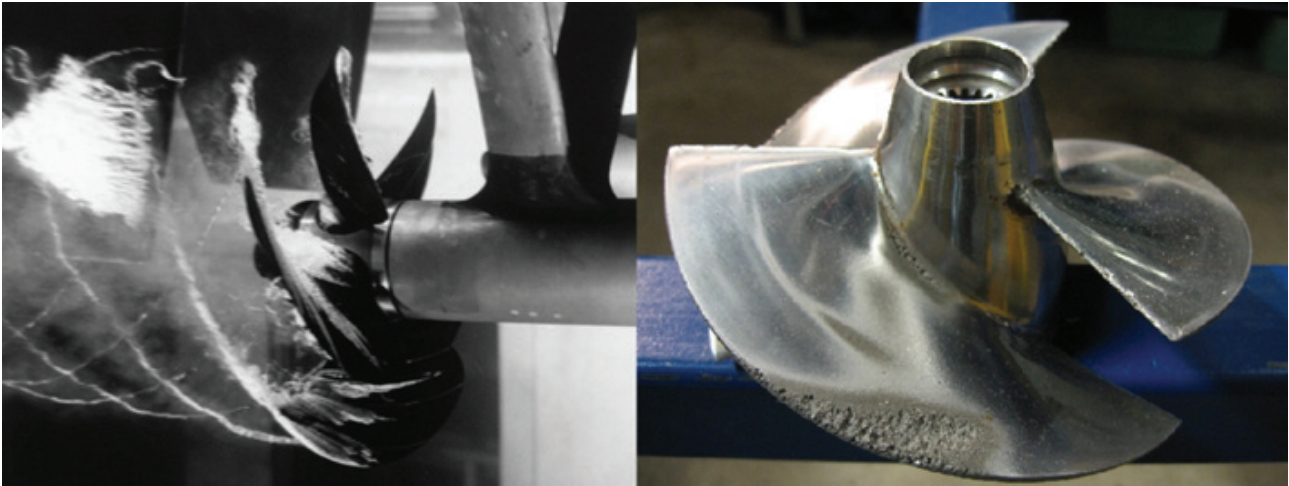


Figure 3: Damaged Induced by Cavitation on the Surface of a Propeller

large volume and increases the surface area of the liquid fuel, to promote better combustion. CFD simulations may then be used to optimise the shape of the spray or the combustion chamber. In the oil and gas industry, when extracting oil a mixture of oil, water, gas and particulate matter is recovered. It may then be of interest to simulate with multiphase CFD the flow in the pipes for flow assurance. Multiphase flow effects may also come as an undesired consequence of the operation of the equipment, for example when cavitation occurs close to a propeller (Figure 3). This may happen when the propeller speed is such that the liquid pressure drops below the saturated vapour pressure in some regions of the propeller blades. Small air bubbles appear which, when they collapse, can cause pitting on the blades and, over time, cause significant damage as shown in Figure 3. In this case, multiphase flow CFD analysis could support propeller design decisions to avoid cavitation or help investigate optimal operating conditions.

### Multiphase Flow Modelling and Coupling

To reproduce the different behaviour of the phases in simulations, modelling multiphase flow numerically means solving conservation equations for different fluid components with phase exchange terms coupling the different phases for mass, momentum, and energy. Other transported variables: e.g. turbulence in the case of RANS modelling may also be included.

Any numerical model should consider the range of scales of interest, from the size of the domain of interest to the characteristic sizes of the bubbles, droplets or particulates, as well as the type of flow, for example Gas-Liquid, or Gas-Solid, or Liquid-Liquid, and the flow regimes.

So, how should one approach such complex and multi-scales flows?

Taking a macroscopic view, answers to this question are often arrived at by considering whether the flow of interest is separated or dispersed, and whether an Eulerian or a Lagrangian point of view should best be adopted.

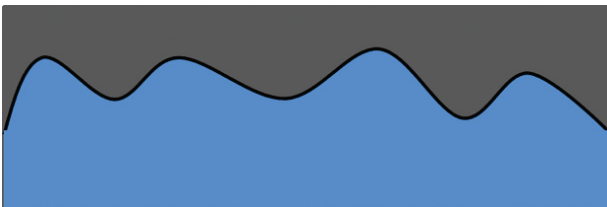


Figure 4: Schematic Representation of Two-Phase, Separated Flow. The Two Materials, in Grey and Blue, are Separated by an Interface Marked as a Black Line.

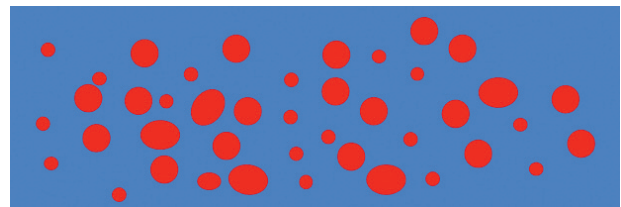


Figure 5: Schematic Representation of Two-Phase Dispersed Flow. The Dispersed Phase is Represented in Red and the Carrier Phase is Represented in Blue.

Copyright: Nicolas Tonello©, Renuda®.



## Separated Flows

When the flow is separated, the different phases occupy different volumes and are separated by interfaces. Leaving the spray aside for the moment, this would be the case, for example, of the ocean waves discussed above. A schematic representation of such a flow is provided in Figure 4.

In this instance, a single fluid framework is often used, whereby the set of conservation equations is solved for a fluid which takes on the different properties of the different fluids in volumes where the fluids are present. For example in the schematic above, imagining that it represents the flow of water in blue and oil in grey, the fluid would have the density of water in the blue regions and the density of oil in the grey regions. To remain consistent with the initial assumption of separated flow, the numerical methods used to carry out the simulation must then be such that a sharp interface is maintained between the fluids. This may be achieved by explicitly tracking the motion of the interface, which can be represented by a set of nodes. Or, the interface may be represented implicitly, for example by computing the relative volumetric proportion of each phase in the domain. Interface capturing is, in general, more applicable because it does not require moving nodes which may become entangled. Two widespread such methods are the Volume of Fluid (VoF) and the Level Set methods.

## Dispersed Flow

When the flow is dispersed, the phases are miscible and interdispersed (Figure 5). For example, this would be the case of the ocean spray made of droplets dispersed in the wind in the photograph shown in Figure 2 or the fuel sprayed into the combustion chamber of an internal combustion engine. Often, the term carrier phase is used to describe the background phase in which the other phase is dispersed.

The coupling between the phases then also involves the momentum exchanges, such as the drag or lift forces, for which closure models must be provided. The system of equations which is solved depends on whether an Eulerian or a Lagrangian point of view is taken. In fully Eulerian approaches, phases are all considered to be continua and a full set of conservation equations may be solved for each phase. The different equations are then coupled through the exchange terms, such as the drag, or the heat transfer terms when phases are not assumed to be in temperature equilibrium. In Eulerian-Eulerian two-phase flow, this would entail solving mass conservation, momentum, and energy equations for each phase. An additional set of equations may also be added to model turbulence. Simplified formulations also exist. For example, in the drift flux framework, the velocity of the drifting phase is computed algebraically from the velocity of the bulk mixture of phases and an assumed drift which describes the local slip between phases. This simplifies the problem by removing the requirement to solve the momentum equations of the dispersed phase.

The alternative approach combines the Eulerian and Lagrangian representations. In the Eulerian-Lagrangian framework, the flow field of the carrier phase is solved using an Eulerian representation on a computational mesh. This means that the velocity, volume fraction, temperature, etc. of the carrier phase is calculated at every computational cell in the mesh, yielding a field of data over the entire computational domain. However, the motion of the dispersed phase is calculated by solving the equations in a Lagrangian framework, which involves following the individual dispersed elements as they move in space. Each element has its own data, such as velocity and temperature, which is updated at each time step. By following individual elements, very fine and very specific physics models may be applied to each element. For example, for the flow of the ocean droplets, one may compute the variation of temperature within individual droplets, or apply a model for the breakup of the individual droplets.

In an Eulerian-Lagrangian simulation, it is usually computationally prohibitive to track the motion of every single physical dispersed element in a phase. Imagine attempting to compute the trajectory of every sand particle in the volcanic eruption. These could quickly number in the tens of millions and more. Instead, a weighting factor may be applied, whereby one numerical particle represents several dispersed elements. The equations are then solved for the numerical particles, which reduces the size of the simulations. However, to compute the coupling terms, the information held at each discrete element's location must be interpolated back onto the continuous Eulerian field for the carrier phase and the information from the Eulerian field must be interpolated at the location of every numerical particle for the dispersed phase. Therefore, the number of numerical particles must still be large enough to be statistically significant.

Because such multi-phase flows can involve a large number of unknowns, and a wide range of physical and time scales, the computational cost (size and duration of calculations) of multi-phase flow modelling can quickly become significantly larger than single phase flows. Therefore, a pragmatic and considered approach is always advisable when simulating such flows. Amongst other things, this applies to coupling, which can sometimes be simplified as we explain in the following section.

## One and Two-way/Four-way Coupling

So far, we have described the different frameworks and how the different phases interact with each other, assuming that all the phases present influence one another and are influenced by one another. This would be termed two-way coupling. Or, four-way coupling if inter-particle collisions are also accounted for. In this case, equations for the different phases should be solved concurrently in time. However, in dispersed flows, there are situations where particles are strongly influenced by



the carrier phase but the discrete phase loading is small enough so that it may be considered to have a negligible influence on the carrier phase. This one-way coupling then opens the door to different solution methodologies, where the carrier phase flow calculation can be decoupled from the dispersed phase flow. Not only can this considerably reduce the computational expense of calculations but it can also significantly speed up parametric studies. In steady-state, one may first compute the flow of the carrier phase to convergence. This may then be used as a 'frozen-flow' field on which dispersed phases may be injected. If phase characteristics need to be changed afterwards, for example the diameter distributions or the material properties, further simulations of the phase fields can then be carried out without changing the carrier phase field.

## Conclusion

This article has provided a brief, overview of the fundamentals of multiphase flow modelling and the coupling between the phases. For clarity, the different concepts have been presented individually, however, in real problems, these would often be encountered together. For example, the images of the waves clearly indicate the presence of both separated, air-water flow in the form of the water waves, and dispersed flow in the form of the sprays. The sprays would result from breakup, which would provide additional coupling terms. Therefore, for the more complex problems, combinations of the approaches described would be used. For a deeper introduction to the subject matter, readers are referred to the fundamental reviews of Ishii and Hibiki [1] and Minier [2]. ■

## References

- [1] M. Ishii and T. Hibiki, Thermo-Fluid Dynamics of Two-Phase Flow, Springer-Verlag New York, 2011
- [2] J.-P. Minier, Statistical Descriptions of Polydisperse Turbulent Two-Phase Flows, Physics Reports, Vol. 665, Elsevier, 15 December 2016

Following a PhD Aerospace Engineering at The University of Michigan, Ann Arbor, MI, USA, specialising in detonations and supersonic reactive flow, **Dr Tonello** worked in the USA for several years developing multiphase, reactive flow CFD software for aerospace applications. After moving to the UK to take up a position leading the implementation of two-phase flow Eulerian modelling and radiation for a leading CFD software provider, Dr Tonello founded Renuda ([renuda.com](http://renuda.com)) in the UK and is now director of Renuda in the UK and France, which specialises in developing software applications and conducting fluid flow and thermal studies. Dr Tonello has 20+ years' experience in CFD and fluid mechanics applied to a large variety of industrial problems and flow regimes, from nuclear to the process industry, with a keen interest and expertise in High Performance Computing (HPC).

Dr Tonello also presented a NAFEMS webinar in 2017 on the topic of Multiphase Flow Modelling, and the recording can be viewed here [nafe.ms/2Dil2Xn](https://nafe.ms/2Dil2Xn)