SIEMENS

Ingenuity for life

Siemens Digital Industries Software

Gaining greater value from CFD data

Speeding up high-fidelity simulations of real-world multiphase flows

Executive summary

This white paper looks at recent advances in multiphase modeling methods in computational fluid dynamics (CFD) and how these improvements are enabling engineers working in the oil and gas industry to capture more physical multiphase flow behaviors in their simulations.

By combining multiphase models, we examine how to extend stratified flow simulations to capture phenomena such as liquid films and droplet transport. This allows us to increase the fidelity of CFD simulations without overburdening computing resources.

In addition, we look at the growing capability of individual multiphase models to capture multiphase flow regimes in a single simulation, focusing on developments in the Eulerian multiphase model, which enables CFD simulations of systems involving stratified regions and slug flows, as well as bubbly and dispersed flow regimes.

Contents

Abstract

Developments in multiphase flow modeling are enabling CFD practitioners to go beyond what has typically been deemed possible, not only creating higher fidelity flow simulations but also extending the application of such simulation approaches to new areas of equipment design and system operations.

CFD is a simulation technique based on the governing equations of fluid flow (Navier-Stokes equations). It offers engineers and scientists the ability to predict fluid flows, heat transfer and other phenomena, including transport of solid materials, chemical reactions and fluid-structure interactions.

CFD has become widely used across the oil and gas industry, from simulations in the wellbore, subsea production systems and pipelines, to the production facility and throughout many applications in the downstream and refining process.

Multiphase flows exist in many situations relating to oil and gas production and processing, and these need to be simulated to aid improved performance, efficiency and reliability. However, multiphase flows are not simple to understand, measure or predict in either the physical or digital/virtual world, and different scenarios require the use of different models.

Multiphase flows

Multiphase flows occur in many different flow regimes, depending on the system through which they are flowing, the flow rates (and speeds), fluid properties and the relative volume fractions of different phases present at any point in the system.

In figure 1 we see some characteristic multiphase flow regimes for horizontal systems, and a reproduction of the widely recognized Baker chart commonly used by engineers who wish to make an initial assessment of potential flow regimes in multiphase pipe flows.

Although we can characterize multiphase flows broadly into regimes, the reality tends to be even more complex and less well-defined, with development of different regimes often changing in time and space within a single system.

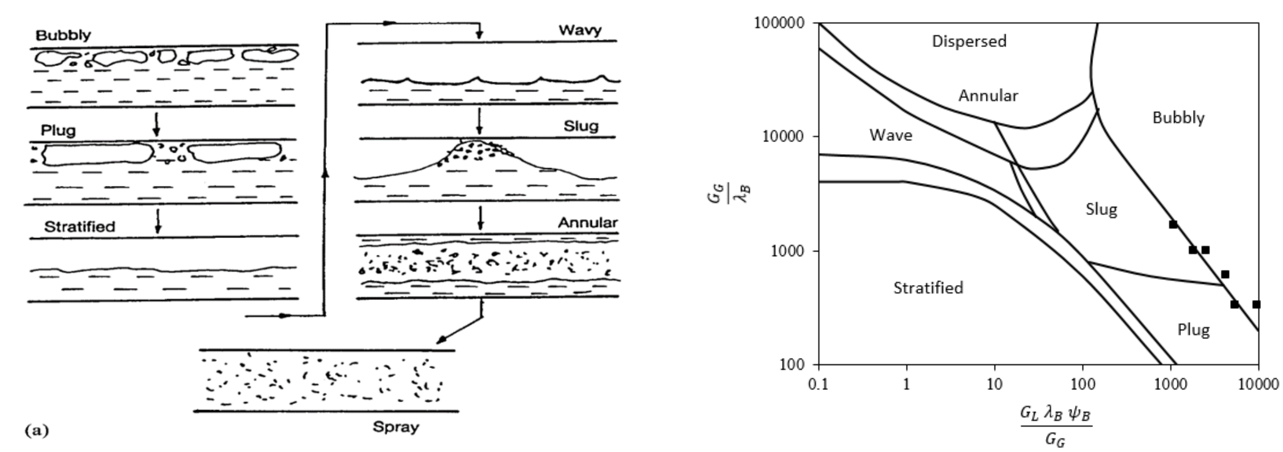


Figure 1. Image showing multiphase flow regimes in horizontal flow and an example of a Baker chart.

Multiphase modeling approaches in CFD

Many approaches have been developed in CFD to model the complexity of multiphase flows. These approaches and their variants are described below.

Eulerian

In the Eulerian multiphase approach, each phase is treated as a continuous phase and the governing flow equations (mass, momentum and energy) are solved along with the volume fraction (portion of volume occupied by a given phase) everywhere within the region simulated. There are several methodologies based on the Eulerian approach:

Eulerian multiphase model

The Eulerian multiphase (EMP) model solves the governing flow equations everywhere in the simulation domain, with volume fraction also being solved to account for changes in the amount of each phase. This concept is termed interpenetrating continua. It is well-suited to a broad range of multiphase flows, but without further augmentation it is unable to be used to predict sharp interfaces between fluid phases.

Multiphase applications where this approach is most appropriate are manifold and include bubbly flows (such as those present in gas flotation used in produced water treatment), mixing and separation, settling applications, slurries and transport of other solids as well as fluidized beds.

Volume of fluid

The volume of fluid (VOF) Eulerian method is applicable to simulating immiscible fluids and capturing the interfaces between them. It is widely applied to free-surface predictions and stratified flows.

The VOF approach can be used to simulate any length scale of free surface, such as the sea and waves in offshore applications, the gas-liquid interface in a separator or even the surface of a droplet in a gas stream.

However, the simulation needs to adequately resolve the interface. This means if we are resolving the interface between gas and liquids in a separator, our model resolution (of a length scale of a few tens of millimeters) is unlikely to be capable of resolving small droplets that may be entrained in the gas flow (of length scale in the micron range). The computational resources (mesh resolution) required would be impractical, even though in theory the VOF method could capture it.

VOF is widely applied to offshore and marine applications to simulate wave loading and wave-structure interactions; separation processes and subsea and pipeline multiphase flows. The standard VOF framework can be enhanced with submodels like cavitation and waves and can be used to study the impact of surface tension on interface stability and flow.

Fluid film model

The fluid film model is also based on the Eulerian approach and is designed to be a computationally efficient approach to capture thin liquid films. If liquid films form in cyclones or inlet devices of separator equipment, they enable liquid droplets to be stripped off into gas flows.

Lagrangian multiphase models

In this method the governing equations of fluid flow (Navier-Stokes equations) are solved for a continuous phase (which could be gas or liquid) and then discrete particles are represented, passing through the main carrier phase(s). The particles can be solid, gas (in liquid) or liquid droplets (in gas).

The Lagrangian approach is well-suited to situations where the volume fraction of the dispersed phase (particles) is low relative to the carrier phase and where interaction of particles with solid boundaries is important.

This approach is widely used to understand transport of liquid droplets in a gas space; for example, within a separator or for sand transport and erosion, where particle impact with walls needs to be captured.

Limitations or challenges are faced in scenarios where a high-volume fraction of particles may be important, perhaps where droplets combine to form liquid pools or films or where high sand fractions build up.

Discrete element method

The discrete element method (DEM) is an extension of the Lagrangian framework. It is used to simulate solid particles and granular material behaviors when we need more detail on the particle-particle and particle-wall contacts than we can resolve from just the Lagrangian multiphase modeling approach (particle tracking).

In the DEM, each particle is resolved fully and can be applied where particle-particle, fluid-particle and particle-wall interactions are important. In addition, it can capture build-up of particles. Since each particle is resolved, the computational expense of this approach increases as the number of particles increases. However, there are methods to reduce computational expense, including clumping or combining several particles together.

Applications in oil and gas can include high-solids loadings in sand transport and erosion, drilling (where particles can be bonded with given strengths), drill-cutting transport as well as solids accumulation and fluidized beds used in the refining process.

Applications and challenges of multiphase CFD

Why are there so many models? Each multiphase model is built on a different set of assumptions in order to address the varying behaviors of flow regimes and the requirements of specific applications.

This works well in cases where one multiphase flow regime dominates. In this section we'll look at two examples: separation performance and liquid entrainment.

Separation performance

CFD is widely used in separator sloshing control for equipment located on floating production units.

In this case it is important to configure internal structures to control sloshing and quantify the loads put on them during vessel motion to aid mechanical design or assess remaining life. Here the loads on the internals are induced by sloshing as a floating production facility moves due to wave action.

In this instance, capturing the interface (free surface) between gas space and liquids is the key to understanding sloshing behavior. We must apply a multiphase CFD simulation model suited to this flow regime – in this case the VOF model.

Figure 2 shows two simulations of the same separator with and without sloshing control baffles. In the left figure there is less control of the liquid interfaces as no baffles are present. However, the right-hand image shows interfaces that are less disturbed due to the presence of internal baffles (which are not visible in the image).

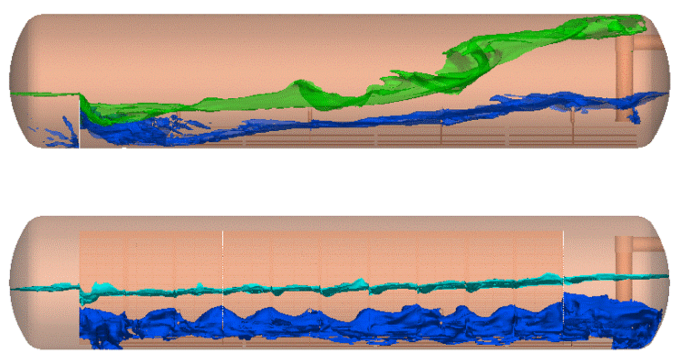


Figure 2. CFD simulation of a separator assessing sloshing performance using VOF approach.

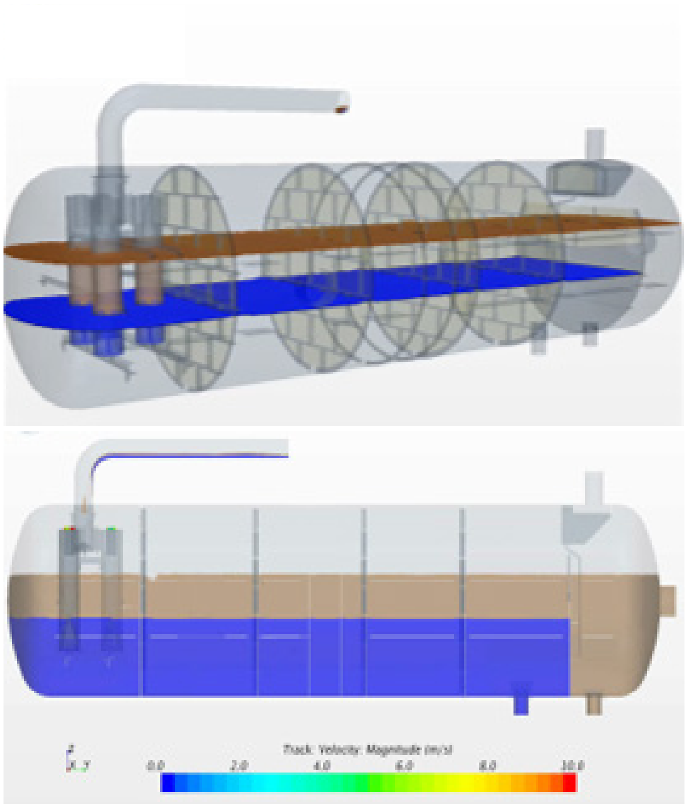


Figure 3. CFD simulation of separator performance simulation with range of multiphase flow regimes being captured.

However, if a separator is in operation and we need to quantify its separation performance, it may be important to capture multiple flow regimes. The vessel in figure 3 (below) shows a separator where CFD is being used to predict performance, which may be carried out during design or perhaps for an upgrade.

In this simulation we have upstream pipework with developing multiphase flows, cyclones where liquid films and droplets are present, gas-liquid and liquid-liquid interfaces and droplets transporting through the gas space.

The challenge in a CFD simulation is knowing which flows to capture, and which will dominate the vessel performance, as well as knowing how to resolve the different behaviors and flow regimes. In a case like this, several stages may be required, which could be:

- Characterize the flow entering the vessel, which may not be straightforward. If the flow regime develops in upstream pipework, it may change as it enters the vessel
- Predict the performance of the inlet device; here the inlet device is a group of cyclones (this could use one of several simulation approaches depending on fluid phase fractions and conditions)
- Assess the bulk separation performance of the vessel capturing the interface behavior between gas and liquid and the liquid-liquid interface, often using the VOF approach
- Investigate liquid carryover potential using Lagrangian techniques

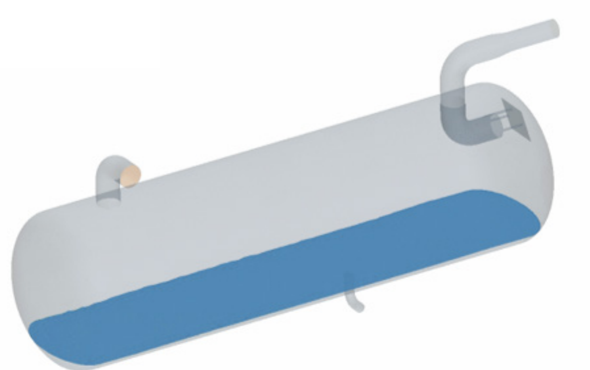
CFD practitioners and engineers can gain critical insights for design and operation by assessing the vessel using each of the steps above. But clearly, any opportunity to combine multiple simulation approaches will enhance accuracy and insights and reduce simulation time.

Entrainment of liquids in gas streams

There are many scenarios where a CFD simulation may need to capture multiple multiphase flow regimes at different length and time scales. For example, when we wish to investigate the potential for liquid entrainment in a gas stream, say from a separator or flare drum during normal operations, or to assess increased risk identified during a process blowdown.

Take the situation where high liquid levels in a vessel lead to elevated gas speeds across the liquid interface. Assessing liquid entrainment in this operating scenario is common during design of a separator. In such scenarios, in the CFD simulation we may need to decide whether to either:

A) Fully resolve the interface and perform transient simulation to capture interface breakup and liquid entrainment using a VOF model. This approach is possible but can be computationally demanding especially in the case of large vessels, long duration transient events or when small liquid droplets develop.



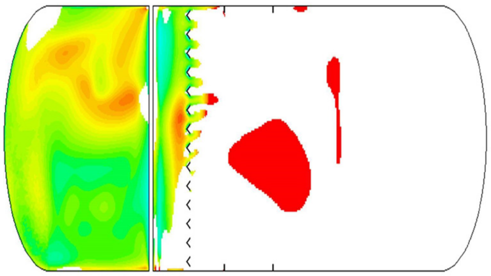


Figure 4. Two simulation approaches for predicting liquid entrainment in process equipment.

B) Make simplifying assumptions and solve the gas flow over the liquid interface, then use empirical relationships to estimate areas of entrainment (perhaps using Helmholtz instability criteria) and potential liquid droplet sizes that may be formed (using relationships like maximum stable droplet size).

This approach requires several assumptions and empirical relationships combined with multiple stages of CFD simulation. It provides a good indication of liquid entrainment potential and demonstrates whether droplets may be entrained into the gas stream. However, it does not explicitly capture the liquid breakup or mass exchange, and so cannot confirm volumes of entrainment or rate of entrainment.

Hybrid multiphase modeling

As referenced above, each multiphase model is built on certain assumptions and is typically suited to specific applications and flow conditions. This is acceptable when applying CFD to a particular flow regime, but it provides a significant constraint or limitation in cases where there is more than one flow regime or where its characterization is the aim of the simulation.

For example, VOF can be applied to capture a separator gas-liquid interface, giving great insight into separator flows and performance. But if we need to assess liquid interface breakup and liquid entrainment, the VOF approach cannot provide the level of fidelity needed to capture the full range of droplet sizes that may be produced. It is too computationally intensive to do this in a single simulation.

However, the development of hybrid multiphase models is beginning to enable us to address such problems and allow a wider range of flow regimes to be captured within a single simulation. The aim of hybrid multiphase modeling is to:

- Enable engineers to apply multiphase CFD models to more flow regimes in a single model – this opens up more applications for simulation to aid design and operations
- Improve the fidelity and resolution of CFD models for complex multiphase flows
- Minimize computational cost and resources needed while increasing model fidelity
- Deploy CFD techniques more quickly to a wider range of technical challenges
- Reduce the number of modeling assumptions needed to apply CFD to multiphase flows
- Ultimately enable greater insight into CFD simulation for more people

Combining multiphase models

Consider a multiphase flow scenario where a liquid flow impinges on a wall. This could be the case of a multiphase flow entering a separator inlet device, for example. When the liquid jet impinges the wall, it may do several things; it may form a thin film (much thinner than the jet itself), or it may break up into droplets rebounding from the wall.

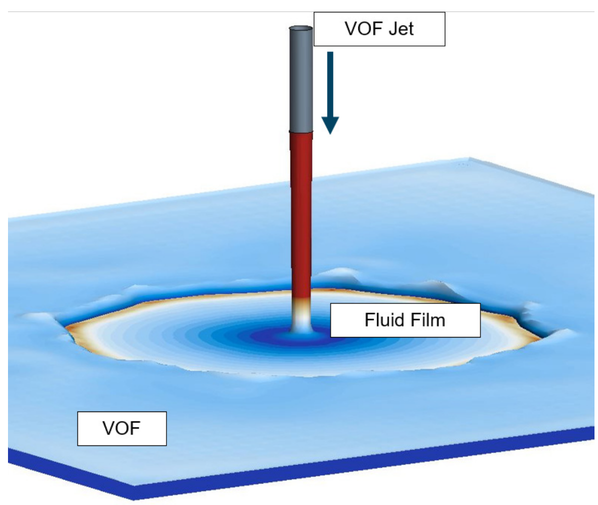


Figure 5. Combining VOF and fluid film models to capture jet impingement.

If we take a hybrid approach to combine the VOF method and the fluid film approach we can use the most appropriate and efficient method in the zone where it is best-suited.

In this case the hybrid multiphase approach has two-way transitions where the VOF jet transitions to a thin fluid film at initial impingement. The film, modeled by the fluid film approach, transitions back to VOF at the hydraulic jump as it becomes too thick for the fluid film approach to resolve. The hydraulic jump, where we transition between the fluid film and VOF approach, is an additional feature required in this hybrid model, which can be seen to be resolved in figure 5.

Hybrid simulation including liquid breakup

In the case demonstrated above in figure 5, no liquid breakup was considered. If a liquid jet impacts a surface, for example, the formation of droplets and liquid rebound is likely. If you are taking a hybrid approach to capture all flow regimes, this requires additional flow phenomena to be captured, for example liquid jets, films and liquid breakup into droplets as well as the subsequent behavior of each.

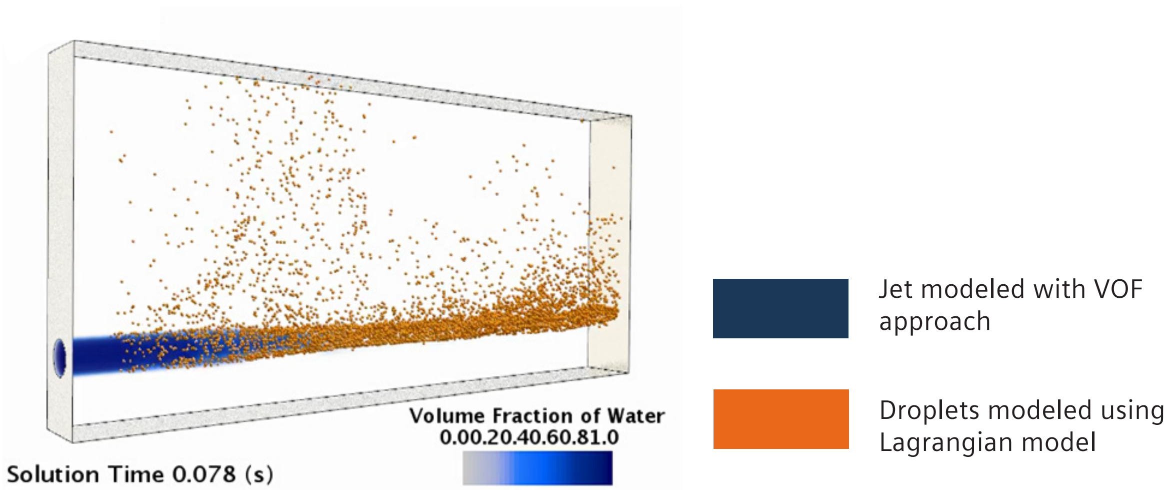


Figure 6. Water jet simulation where VOF is used to resolve the jet core and the Lagrangian approach to resolve the stripped-off droplets.

The simulation was conducted using Simcenter™ STAR-CCM+™ software, which is a part of the Xcelerator™ portfolio, the comprehensive and integrated portfolio of software and services from Siemens Digital Industries Software.

In figure 6 we have a liquid jet breaking up. The core of the jet is a continuous water phase with liquid droplets stripped off the surface by the shear layer between the fast-moving jet and surrounding air.

Using a VOF approach alone to capture the jet and droplets formed would require a computationally intensive simulation. In many cases it would be impractical to capture the droplet formation.

Using a hybrid approach, the VOF model is used to capture the core water jet, and the Lagrangian model is used to simulate the formation of the droplets, which occur at length scales below the resolution of the VOF model.

This saves computational expense and time, while enabling all multiphase flows to be simulated.

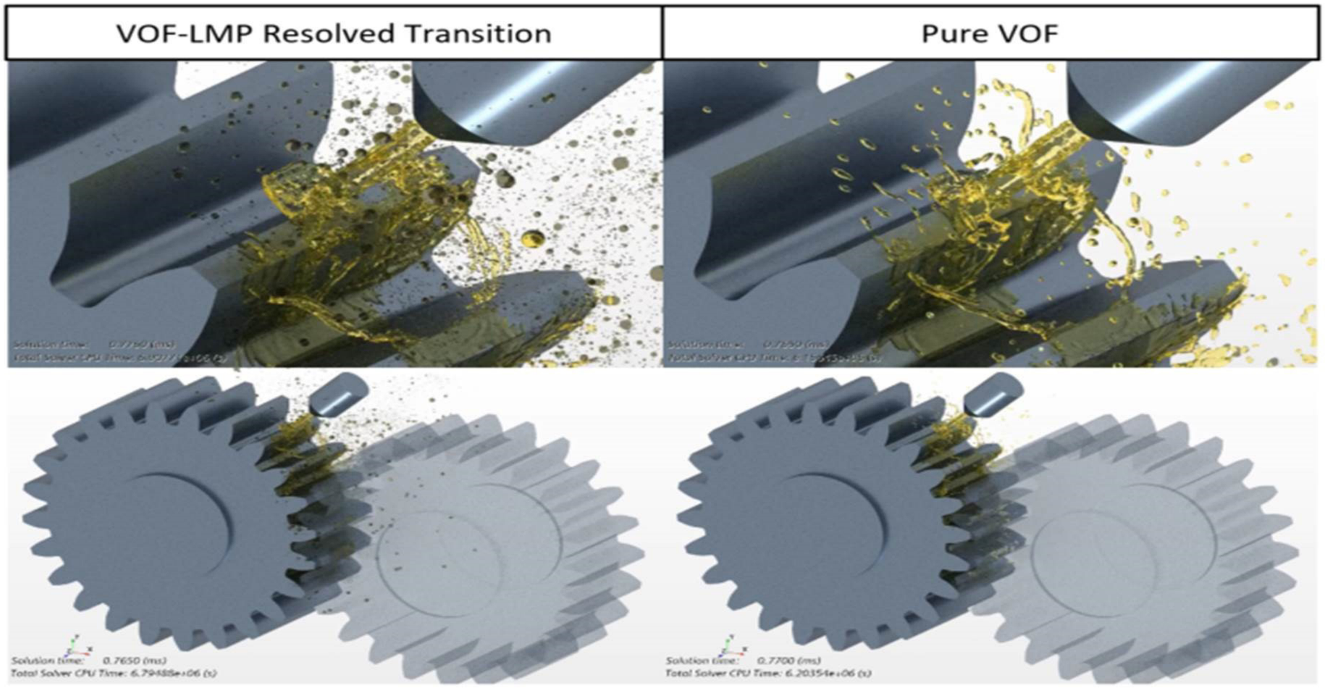


Figure 7. Oil jet impacting on rotating gear system.

In figure 7, we see an oil jet impacting on rotating gear system. The image on the right shows a simulation where a VOF approach is used to resolve all flow features from the jet to liquid films, droplet formation and transport. This requires a highly resolved numerical mesh. The image on the right shows a simulation of the same scenario. However, the jet is resolved using the VOF approach the droplet formation and transport is solved using the Lagrangian model. While both simulations solve the same flow phenomena, the simulation depicted on the left requires significantly less computational resource.

Expanded capability of multiphase models

A further approach to hybrid multiphase modeling in CFD is to extend the core models so they may be applied to even more multiphase flow regimes rather than by combining models as described above.

For example, there are many industrial applications where different flow regimes may be present, or dominate, different parts of the same system. Consider a gas flotation system, often used in separating oil from produced water. In such systems free surface behaviors may dominate at the interface between water and separated oil layer; this is well suited to a VOF approach. However, within the region of the vessel where flotation gas columns exist, the bubbly flow regime may dominate and a Eulerian multiphase model will be more appropriate.

In such cases, a compromise is traditionally required depending on which behavior is considered most important to the question being investigated. A recent development of the Eulerian multiphase modeling approach aims to address this issue; the large-scale interface (LSI) and multiple regime model (MRM).

Large-scale interface and multiple regime models

Termed the Eulerian multiphase model with large-scale interface (LSI) and multiple flow regime models, these approaches enable simulation of a scenario with multiple multiphase flow regimes present in a single system, as set out below in figure 8.

Here we have multiple multiphase flow regimes present in a single system containing two phases, A and B. To the left of the figure we have several different zones exhibiting the dispersed flow regime. To the upper left

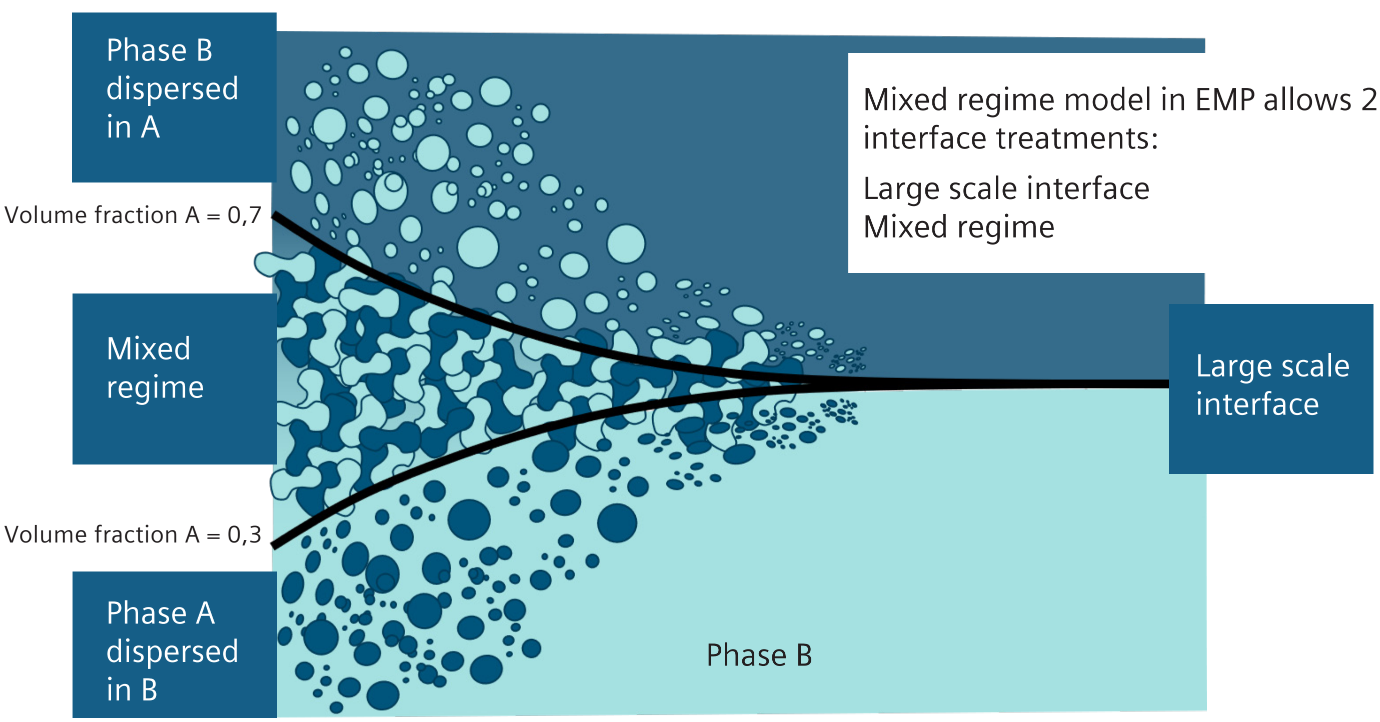


Figure 8. Eulerian multiphase model with large-scale interface and mixed regime model to capture multiple regimes.

we have phase B dispersed in phase A, in the lower left we have phase A dispersed in phase B. To the center left of figure 8 we have a mixed regime where the volume fractions of each phase are more equal. To the right of the figure a clear interface forms and we have a stratified flow regime. This is clearly a complex and changing scenario.

A VOF approach would resolve the stratified zone, but it would require excessive computational resources to resolve the zones of dispersed phases or mixing. A standard Eulerian multiphase model may capture one dispersed zone but would be challenged in the other dispersed zone (due to the formulation of which phase is defined to exist within the other) and would not capture a sharp interface in the stratified zone.

The introduction of the LSI approach enables the Eulerian multiphase model to capture the interface, while the multiple regime model allows the different mixed and dispersed flows to also be captured in a single simulation.

The extended capability of multiphase models is helping to expand the number of applications and remove barriers to the applicability of CFD across complex processing and multiphase applications.

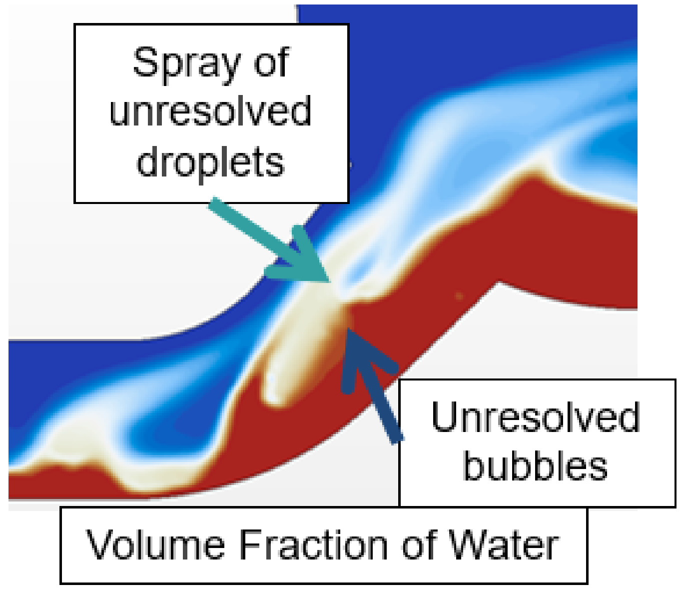


Figure 9. Snapshot in time of showing capture of regions of stratification, liquid spray and bubbly regions.

Conclusion

Developments in model capability and simulation speed are enabling us to gain more confidence in the data we obtain from CFD simulations and allows delivery of high-fidelity simulations of real-world multiphase flows in less time.

Ultimately, these advances in CFD techniques help designers and operators of multiphase systems gain greater value from CFD simulation and obtain greater insight into issues relating to fluid dynamics. This in turn contributes to greater safety, efficiency and cost effectiveness in oil and gas industry operations.

Siemens Digital Industries Software

Headquarters
Granite Park One
5800 Granite Parkway
Suite 600
Plano, TX 75024
USA
+1 972 987 3000

Americas
Granite Park One
5800 Granite Parkway
Suite 600
Plano, TX 75024
USA
+1 314 264 8499

Europe
Stephenson House
Sir William Siemens Square
Frimley, Camberley
Surrey, GU16 8QD
+44 (0) 1276 413200

Asia-Pacific
Unit 901-902, 9/F
Tower B, Manulife Financial Centre
223-231 Wai Yip Street, Kwun Tong
Kowloon, Hong Kong
+852 2230 3333

About Siemens Digital Industries Software

Siemens Digital Industries Software is driving transformation to enable a digital enterprise where engineering, manufacturing and electronics design meet tomorrow. Xcelerator, the comprehensive and integrated portfolio of software and services from Siemens Digital Industries Software, helps companies of all sizes create and leverage a comprehensive digital twin that provides organizations with new insights, opportunities and levels of automation to drive innovation. For more information on Siemens Digital Industries Software products and services, visit siemens.com/software or follow us on LinkedIn, Twitter, Facebook and Instagram. Siemens Digital Industries Software – Where today meets tomorrow.

siemens.com/software

83711-C4 7/21 K