Chapter 17: Multiphase Flows

This chapter discusses the general multiphase models that are available in ANSYS Fluent. Introduction (p. 465) provides a brief introduction to multiphase modeling, Discrete Phase (p. 373) discusses the Lagrangian dispersed phase model, and Solidification and Melting (p. 601) describes ANSYS Fluent's model for solidification and melting. For information about using the general multiphase models in ANSYS Fluent, see Modeling Multiphase Flows in the User's Guide. Information about the various theories behind the multiphase models is presented in the following sections:

17.1. Introduction

A large number of flows encountered in nature and technology are a mixture of phases. Physical phases of matter are gas, liquid, and solid, but the concept of phase in a multiphase flow system is applied in a broader sense. In multiphase flow, a phase can be defined as an identifiable class of material that has a particular inertial response to and interaction with the flow and the potential field in which it is immersed. For example, different-sized solid particles of the same material can be treated as different phases because each collection of particles with the same size will have a similar dynamical response to the flow field.

Information is organized into the following subsections:

17.1.1. Multiphase Flow Regimes

Multiphase flow regimes can be grouped into four categories: gas-liquid or liquid-liquid flows; gas-solid flows; liquid-solid flows; and three-phase flows.

17.1.1.1. Gas-Liquid or Liquid-Liquid Flows

The following regimes are gas-liquid or liquid-liquid flows:

- Bubbly flow: This is the flow of discrete gaseous or fluid bubbles in a continuous fluid.
- Droplet flow: This is the flow of discrete fluid droplets in a continuous gas.
- Slug flow: This is the flow of large bubbles in a continuous fluid.
- Stratified/free-surface flow: This is the flow of immiscible fluids separated by a clearly-defined interface.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

See Figure 17.1: Multiphase Flow Regimes (p. 467) for illustrations of these regimes.

17.1.1.2. Gas-Solid Flows

The following regimes are gas-solid flows:

- Particle-laden flow: This is flow of discrete particles in a continuous gas.
- Pneumatic transport: This is a flow pattern that depends on factors such as solid loading, Reynolds numbers, and particle properties. Typical patterns are dune flow, slug flow, and homogeneous flow.
- Fluidized bed: This consists of a vessel containing particles, into which a gas is introduced through a distributor. The gas rising through the bed suspends the particles. Depending on the gas flow rate, bubbles appear and rise through the bed, intensifying the mixing within the bed.

See Figure 17.1: Multiphase Flow Regimes (p. 467) for illustrations of these regimes.

17.1.1.3. Liquid-Solid Flows

The following regimes are liquid-solid flows:

- Slurry flow: This flow is the transport of particles in liquids. The fundamental behavior of liquid-solid flows varies with the properties of the solid particles relative to those of the liquid. In slurry flows, the Stokes number (see Equation 17.4 (p. 471)) is normally less than 1. When the Stokes number is larger than 1, the characteristic of the flow is liquid-solid fluidization.
- Hydrotransport: This describes densely-distributed solid particles in a continuous liquid.
- Sedimentation: This describes a tall column initially containing a uniform dispersed mixture of particles. At the bottom, the particles will slow down and form a sludge layer. At the top, a clear interface will appear, and in the middle a constant settling zone will exist.

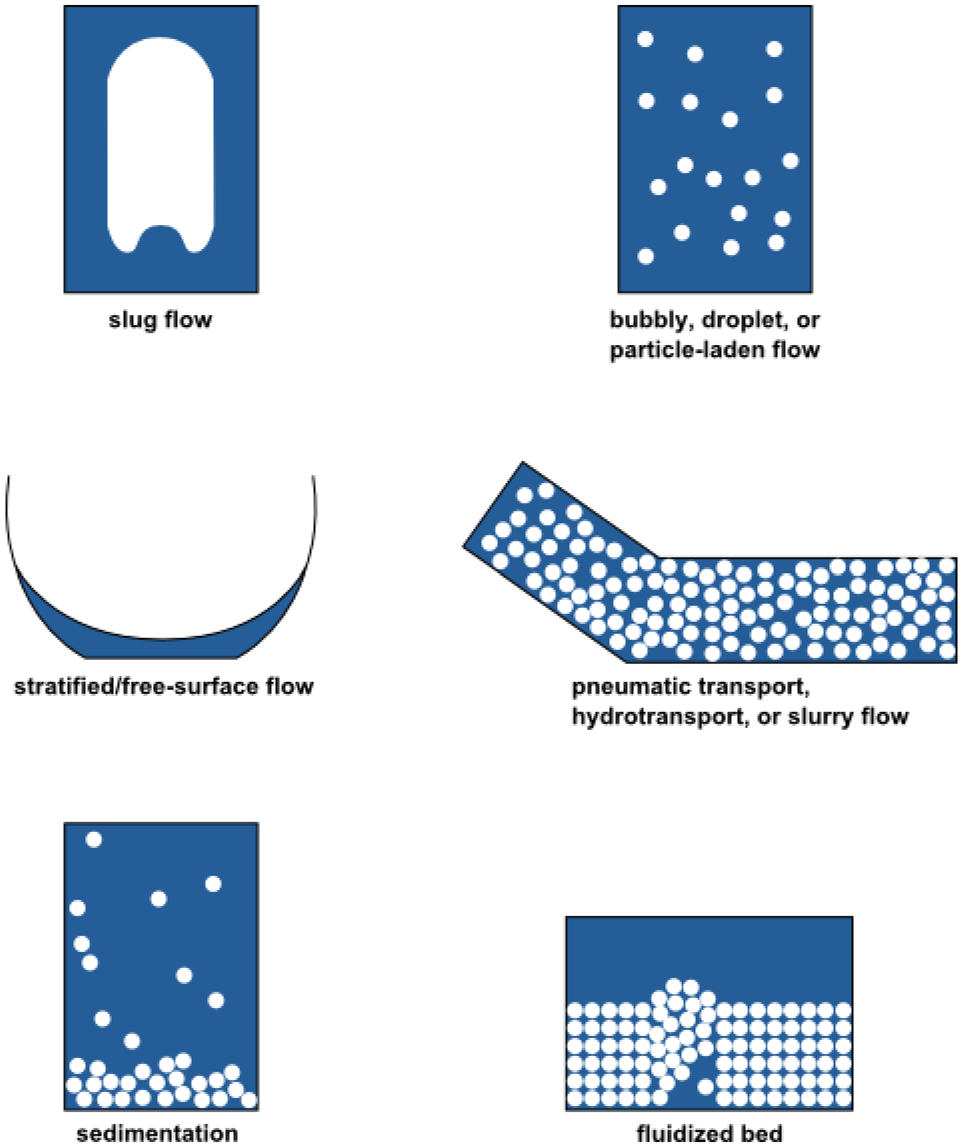
See Figure 17.1: Multiphase Flow Regimes (p. 467) for illustrations of these regimes.

17.1.1.4. Three-Phase Flows

Three-phase flows are combinations of the other flow regimes listed in the previous sections.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 17.1: Multiphase Flow Regimes



17.1.2. Examples of Multiphase Systems

Specific examples of each regime described in Multiphase Flow Regimes (p. 465) are listed below:

- Bubbly flow examples include absorbers, aeration, air lift pumps, cavitation, evaporators, flotation, and scrubbers.
- Droplet flow examples include absorbers, atomizers, combustors, cryogenic pumping, dryers, evaporation, gas cooling, and scrubbers.
- Slug flow examples include large bubble motion in pipes or tanks.
- Stratified/free-surface flow examples include sloshing in offshore separator devices and boiling and condensation in nuclear reactors.
- Particle-laden flow examples include cyclone separators, air classifiers, dust collectors, and dust-laden environmental flows.

- Pneumatic transport examples include transport of cement, grains, and metal powders.
- Fluidized bed examples include fluidized bed reactors and circulating fluidized beds.
- Slurry flow examples include slurry transport and mineral processing
- Hydrotransport examples include mineral processing and biomedical and physiochemical fluid systems
- Sedimentation examples include mineral processing.

17.2. Choosing a General Multiphase Model

The first step in solving any multiphase problem is to determine which of the regimes described in Multiphase Flow Regimes (p. 465) best represents your flow. Model Comparisons (p. 469) provides some broad guidelines for determining appropriate models for each regime, and Detailed Guidelines (p. 470) provides details about how to determine the degree of interphase coupling for flows involving bubbles, droplets, or particles, and the appropriate model for different amounts of coupling.

Information is organized into the following subsections:

17.2.1. Approaches to Multiphase Modeling

Advances in computational fluid mechanics have provided the basis for further insight into the dynamics of multiphase flows. Currently there are two approaches for the numerical calculation of multiphase flows: the Euler-Lagrange approach (discussed in Introduction (p. 373)) and the Euler-Euler approach (discussed in the following section).

17.2.1.1. The Euler-Euler Approach

In the Euler-Euler approach, the different phases are treated mathematically as interpenetrating continua. Since the volume of a phase cannot be occupied by the other phases, the concept of phasic volume fraction is introduced. These volume fractions are assumed to be continuous functions of space and time and their sum is equal to one. Conservation equations for each phase are derived to obtain a set of equations, which have similar structure for all phases. These equations are closed by providing constitutive relations that are obtained from empirical information, or, in the case of granular flows , by application of kinetic theory.

In ANSYS Fluent, three different Euler-Euler multiphase models are available: the volume of fluid (VOF) model, the mixture model, and the Eulerian model.

17.2.1.1.1. The VOF Model

The VOF model (described in Volume of Fluid (VOF) Model Theory (p. 474)) is a surface-tracking technique applied to a fixed Eulerian mesh. It is designed for two or more immiscible fluids where the position of the interface between the fluids is of interest. In the VOF model, a single set of momentum equations is shared by the fluids, and the volume fraction of each of the fluids in each computational cell is tracked throughout the domain. Applications of the VOF model include stratified flows, free-surface flows, filling, sloshing, the motion of large bubbles in a liquid, the motion of liquid after a dam break, the prediction of jet breakup (surface tension), and the steady or transient tracking of any liquid-gas interface.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

17.2.1.1.2. The Mixture Model

The mixture model (described in Mixture Model Theory (p. 500)) is designed for two or more phases (fluid or particulate). As in the Eulerian model, the phases are treated as interpenetrating continua. The mixture model solves for the mixture momentum equation and prescribes relative velocities to describe the dispersed phases. Applications of the mixture model include particle-laden flows with low loading, bubbly flows, sedimentation , and cyclone separators. The mixture model can also be used without relative velocities for the dispersed phases to model homogeneous multiphase flow.

17.2.1.1.3. The Eulerian Model

The Eulerian model (described in Eulerian Model Theory (p. 511)) is the most complex of the multiphase models in ANSYS Fluent. It solves a set of n momentum and continuity equations for each phase. Coupling is achieved through the pressure and interphase exchange coefficients. The manner in which this coupling is handled depends upon the type of phases involved; granular (fluid-solid) flows are handled differently than nongranular (fluid-fluid) flows. For granular flows, the properties are obtained from application of kinetic theory. Momentum exchange between the phases is also dependent upon the type of mixture being modeled. ANSYS Fluent's user-defined functions allow you to customize the calculation of the momentum exchange. Applications of the Eulerian multiphase model include bubble columns, risers, particle suspension, and fluidized beds.

17.2.2. Model Comparisons

In general, once you have determined the flow regime that best represents your multiphase system, you can select the appropriate model based on the following guidelines:

- For bubbly, droplet, and particle-laden flows in which the phases mix and/or dispersed-phase volume fractions exceed 10%, use either the mixture model (described in Mixture Model Theory (p. 500)) or the Eulerian model (described in Eulerian Model Theory (p. 511)).

- For slug flows, use the VOF model. See Volume of Fluid (VOF) Model Theory (p. 474) for more information about the VOF model.

- For stratified/free-surface flows, use the VOF model. See Volume of Fluid (VOF) Model Theory (p. 474) for more information about the VOF model.

- For pneumatic transport, use the mixture model for homogeneous flow (described in Mixture Model Theory (p. 500)) or the Eulerian model for granular flow (described in Eulerian Model Theory (p. 511)).

- For fluidized beds, use the Eulerian model for granular flow. See Eulerian Model Theory (p. 511) for more information about the Eulerian model.

- For slurry flows and hydrotransport, use the mixture or Eulerian model (described, respectively, in Mixture Model Theory (p. 500), Eulerian Model Theory (p. 511)).

- For sedimentation, use the Eulerian model. See Eulerian Model Theory (p. 511) for more information about the Eulerian model.

- For general, complex multiphase flows that involve multiple flow regimes, select the aspect of the flow that is of most interest, and choose the model that is most appropriate for that aspect of the flow. Note that the accuracy of results will not be as good as for flows that involve just one flow regime, since the model you use will be valid for only part of the flow you are modeling.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

As discussed in this section, the VOF model is appropriate for stratified or free-surface flows, and the mixture and Eulerian models are appropriate for flows in which the phases mix or separate and/or dispersed-phase volume fractions exceed 10%. (Flows in which the dispersed-phase volume fractions are less than or equal to 10% can be modeled using the discrete phase model described in Discrete Phase (p. 373).)

To choose between the mixture model and the Eulerian model, you should consider the following guidelines:

- If there is a wide distribution of the dispersed phases (that is, if the particles vary in size and the largest particles do not separate from the primary flow field), the mixture model may be preferable (that is, less computationally expensive). If the dispersed phases are concentrated just in portions of the domain, you should use the Eulerian model instead.
- If interphase drag laws that are applicable to your system are available (either within ANSYS Fluent or through a user-defined function), the Eulerian model can usually provide more accurate results than the mixture model. Even though you can apply the same drag laws to the mixture model, as you can for a non-granular Eulerian simulation, if the interphase drag laws are unknown or their applicability to your system is questionable, the mixture model may be a better choice. For most cases with spherical particles, the Schiller-Naumann law is more than adequate. For cases with non-spherical particles, a user-defined function can be used.
- If you want to solve a simpler problem, which requires less computational effort, the mixture model may be a better option, since it solves a smaller number of equations than the Eulerian model. If accuracy is more important than computational effort, the Eulerian model is a better choice. Keep in mind, however, that the complexity of the Eulerian model can make it less computationally stable than the mixture model.

ANSYS Fluent's multiphase models are compatible with ANSYS Fluent's dynamic mesh modeling feature. For more information on the dynamic mesh feature, see Flows Using Sliding and Dynamic Meshes (p. 33). For more information about how other ANSYS Fluent models are compatible with ANSYS Fluent's multiphase models, see Appendix A in the User's Guide.

17.2.2.1. Detailed Guidelines

For stratified and slug flows, the choice of the VOF model, as indicated in Model Comparisons (p. 469), is straightforward. Choosing a model for the other types of flows is less straightforward. As a general guide, there are some parameters that help to identify the appropriate multiphase model for these other flows: the particulate loading, β, and the Stokes number, St. (Note that the word "particle" is used in this discussion to refer to a particle, droplet, or bubble.)

17.2.2.1.1. The Effect of Particulate Loading

Particulate loading has a major impact on phase interactions. The particulate loading is defined as the mass density ratio of the dispersed phase (d) to that of the carrier phase (c):

$$\beta=\frac{\alpha_d\rho_d}{\alpha_c\rho_c} \tag{17.1}$$

The material density ratio

$$\gamma=\frac{\rho_d}{\rho_c} \tag{17.2}$$

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

is greater than 1000 for gas-solid flows, about 1 for liquid-solid flows, and less than 0.001 for gas-liquid flows.

Using these parameters it is possible to estimate the average distance between the individual particles of the particulate phase. An estimate of this distance has been given by Crowe et al. [79] (p. 733):

$$\frac{L}{d_d}=\left(\frac{\pi}{6}\frac{1+\kappa}{\kappa}\right)^{1/3} \tag{17.3}$$

where $\kappa=\frac{\beta}{\gamma}$. Information about these parameters is important for determining how the dispersed phase should be treated. For example, for a gas-particle flow with a particulate loading of 1, the interparticle space $\frac{L}{d_d}$ is about 8; the particle can therefore be treated as isolated (that is, very low particulate loading).

Depending on the particulate loading, the degree of interaction between the phases can be divided into the following three categories:

- For very low loading, the coupling between the phases is one-way (that is, the fluid carrier influences the particles via drag and turbulence, but the particles have no influence on the fluid carrier). The discrete phase (Discrete Phase (p. 373)), mixture, and Eulerian models can all handle this type of problem correctly. Since the Eulerian model is the most expensive, the discrete phase or mixture model is recommended.
- For intermediate loading, the coupling is two-way (that is, the fluid carrier influences the particulate phase via drag and turbulence, but the particles in turn influence the carrier fluid via reduction in mean momentum and turbulence). The discrete phase (Discrete Phase (p. 373)), mixture, and Eulerian models are all applicable in this case, but you need to take into account other factors in order to decide which model is more appropriate. See below for information about using the Stokes number as a guide.
- For high loading, there is two-way coupling plus particle pressure and viscous stresses due to particles (four-way coupling). Only the Eulerian model will handle this type of problem correctly.

17.2.2.1.2. The Significance of the Stokes Number

For systems with intermediate particulate loading, estimating the value of the Stokes number can help you select the most appropriate model. The Stokes number can be defined as the relation between the particle response time and the system response time:

$$St=\frac{\tau_d}{t_s} \tag{17.4}$$

where $\tau_d=\frac{\rho_d d_d^2}{18\mu_c}$ and t_s is based on the characteristic length (L_s) and the characteristic velocity (V_s) of the system under investigation: $t_s=\frac{L_s}{V_s}$.

For $St \ll 1.0$, the particle will follow the flow closely and any of the three models (discrete phase(Discrete Phase (p. 373)), mixture, or Eulerian) is applicable; you can therefore choose the least expensive (the mixture model, in most cases), or the most appropriate considering other factors. For $St > 1.0$, the particles will move independently of the flow and either the discrete phase model (Discrete Phase (p. 373)) or the Eulerian model is applicable. For $St \approx 1.0$, again any of the three models is applicable; you can choose the least expensive or the most appropriate considering other factors.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

17.2.2.1.2.1. Examples

For a coal classifier with a characteristic length of 1 m and a characteristic velocity of 10 m/s, the Stokes number is 0.04 for particles with a diameter of 30 microns, but 4.0 for particles with a diameter of 300 microns. Clearly the mixture model will not be applicable to the latter case.

For the case of mineral processing, in a system with a characteristic length of 0.2 m and a characteristic velocity of 2 m/s, the Stokes number is 0.005 for particles with a diameter of 300 microns. In this case, you can choose between the mixture and Eulerian models. The volume fractions are too high for the discrete phase model (Discrete Phase (p. 373)), as noted below.

17.2.2.1.3. Other Considerations

Keep in mind that the use of the discrete phase model (Discrete Phase (p. 373)) is limited to low volume fractions, unless you are using the dense discrete phase model formulation. In addition, for the discrete phase model simulation, you can choose many more advanced combustion models compared to the Eulerian models. To account for particle distributions, you will need to use the population balance models (see the Population Balance Module Manual) or the discrete phase model and the dense discrete phase model.

17.2.3. Time Schemes in Multiphase Flow

In many multiphase applications, the process can vary spatially as well as temporally. In order to accurately model multiphase flow, both higher-order spatial and time discretization schemes are necessary. In addition to the first-order time scheme in ANSYS Fluent, the second-order time scheme is available in the Mixture and Eulerian multiphase models, and with the VOF Implicit Scheme.

Important

The second-order time scheme cannot be used with the VOF Explicit Schemes.

The second-order time scheme has been adapted to all the transport equations, including mixture phase momentum equations, energy equations, species transport equations, turbulence models, phase volume fraction equations, the pressure correction equation, and the granular flow model. In multiphase flow, a general transport equation (similar to that of Equation 20.19 (p. 639)) may be written as

$$\frac{\partial\left(\alpha\rho\phi\right)}{\partial t}+\nabla\cdot\left(\alpha\rho\vec{V}\phi\right)=\nabla\cdot\overline{\overline{\tau}}+S_{\phi} \tag{17.5}$$

Where ϕ is either a mixture (for the mixture model) or a phase variable, α is the phase volume fraction (unity for the mixture equation), ρ is the mixture phase density, $\vec{V}$ is the mixture or phase velocity (depending on the equations), $\overline{\overline{\tau}}$ is the diffusion term, and S_{ϕ} is the source term.

As a fully implicit scheme, this second-order time-accurate scheme achieves its accuracy by using an Euler backward approximation in time (see Equation 20.21 (p. 639)). The general transport equation, Equation 17.5 (p. 472) is discretized as

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$\frac{3\left(\alpha_p\rho_p\phi_p Vol\right)^{n+1}-4\left(\alpha_p\rho_p\phi_p Vol\right)^{n}+\left(\alpha_p\rho_p\phi_p\right)^{n-1}}{2\,\Delta t}= \sum\left[A_{nb}\left(\phi_{nb}-\phi_p\right)\right]^{n+1}+S_U{}^{n+1}-S_p{}^{n+1}\phi_p{}^{n+1} \tag{17.6}$$

Equation 17.6 (p. 473) can be written in simpler form:

$$A_p\phi_p=\sum A_nb\phi_nb+S_\phi \tag{17.7}$$

where

$$A_p=\sum A_{nb}{}^{n+1}+S_p{}^{n+1}+\frac{1.5\left(\alpha_p\rho_p Vol\right)^{n+1}}{\Delta t}$$

$$S_\phi=S_U{}^{n+1}+\frac{2\left(\alpha_p\rho_p\phi_p Vol\right)^{n}-0.5\left(\alpha_p\rho_p\phi_p Vol\right)^{n-1}}{\Delta t}$$

This scheme is easily implemented based on ANSYS Fluent's existing first-order Euler scheme. It is unconditionally stable, however, the negative coefficient at the time level t_{n-1}, of the three-time level method, may produce oscillatory solutions if the time steps are large.

This problem can be eliminated if a bounded second-order scheme is introduced. However, oscillating solutions are most likely seen in compressible liquid flows. Therefore, in this version of ANSYS Fluent, a bounded second-order time scheme has been implemented for compressible liquid flows only. For single phase and multiphase compressible liquid flows, the second-order time scheme is, by default, the bounded scheme.

17.2.4. Stability and Convergence

The process of solving a multiphase system is inherently difficult and you may encounter some stability or convergence problems.

When solving a time-dependent problem, a proper initial field is required to avoid instabilities, which usually arise from poor initial fields. If the CPU time is a concern for transient problems, then the best option is to use PC SIMPLE. When body forces are significant, or if the solution requires higher order numerical schemes, it is recommended that you start with a small time step, which can be increased after performing a few time steps to get a better approximation of the pressure field.

For a steady solution, it is recommended that you use the Multiphase Coupled solver, described in detail in Coupled Solution for Eulerian Multiphase Flows in the User's Guide. The iterative nature of this solver requires a good starting patched field. If difficulties are encountered due to higher order schemes, or due to the complexities of the problem, you may need to reduce the Courant number. The default Courant number is 200 but it can be reduced to as low as 4. This can later be increased if the iteration process runs smoothly. In addition, there are explicit under-relaxation factors for velocities and pressure. All other under-relaxation factors are implicit. Lower under-relaxation factors for the volume fraction equation may delay the solution dramatically with the Coupled solver (any value 0.5 or above is adequate); on the contrary, PC SIMPLE would normally need a low under-relaxation for the volume fraction equation.

Release 15.0 - © SAS IP, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.