Multiphase Laminar Flow with More Than Two Phases

William T. Vetterling*
Zink Imaging
*Corresponding author: 37 Manning Road, Billerica, MA 01821, vetterw@zink.com

Abstract: Two-phase laminar flow and turbulent flow problems may be modeled in COMSOL MP by using the level-set method. For laminar flow problems, we discuss how to extend this capability to problems with more than two fluids. We also introduce an alternate method of solving multiphase flow problems that does not use a level-set function. This method does not currently include the effects of surface tension.

Keywords: Level-Set, Multiphase, Fluid, Flow, Laminar

1. Introduction

The COMSOL Multiphysics Fluid Flow module has a **Laminar Two-Phase Flow** interface for solving problems with the Level Set method.[1] Building upon the capabilities of a Navier-Stokes solver, this interface notably allows the solution of problems in which the fluid regions undergo changes in topology, (e.g. when they divide or coalesce) and in which account must be taken of interfacial tension and contact angles at the boundaries.

This is a powerful modeling tool, but in its native form it is designed for use with just two fluids. In this paper we discuss how the tool may be extended for use in problems having more than two fluids, provided that the flow is laminar. We then introduce a second method capable of solving such problems, but for which no method of including the effects of surface tension has yet been formulated.

2. Level Set Method

The Level Set interface makes use of a smooth auxiliary function $\xi(\vec{r},t)$ that goes from 0 to 1, and has a $\xi = 0.5$ level set that divides the model geometry into regions containing each of the two fluid phases. For example, at any time t, the regions containing fluid #1 are those with $\xi(\vec{r},t) \leq 0.5$, and those containing fluid #2 are those with $\xi(\vec{r},t) > 0.5$. At time t=0, the user provides the solver with an original configuration of the fluids, in the form of a set of boundaries that initially separate the fluids, and a specification of which domains contain each of the fluids. From this information, the solver carries out an initialization step in which it produces a starting Level Set function $\xi(\vec{r},t_0)$ that has $\xi = 0.5$ on the identified boundaries and that correctly identifies the fluids in each domain. As the simulation proceeds, the Navier-Stokes equation is used to compute fluid motion, and a differential equation is applied to $\xi(\vec{r},t)$ so that its $\xi = 0.5$ level set properly tracks the coordinates of the interfacial boundaries. Knowing that the value of ξ must remain 0.5 at the interfacial boundaries, it would appear that a suitable PDE for $\xi(\vec{r},t)$ might be:

$$\frac{d\xi(\vec{r},t)}{dt} = 0 \quad \rightarrow \quad \vec{u}\cdot\nabla\xi + \frac{\partial\xi}{\partial t} = 0$$

This equation moves the level set contours at the local velocity $\vec{u}$ of the fluid. In practice, this equation is found to result in "roughening" of the interfacial boundaries as they propagate. Steps are therefore taken to "reinitialize" ξ, periodically restoring the norm of its gradient at the level-set contour. "Initialization-free" schemes[2, 3] have been developed. COMSOL MP implements a conservative level-set method with a modified equation for the auxiliary function: [4]

$$\frac{\partial\xi}{\partial t} + \nabla\xi\cdot\vec{u} = \gamma\nabla\cdot\left(\epsilon\nabla\xi - \xi(1-\xi)\left(\frac{\nabla\xi}{|\nabla\xi|}\right)\right).$$

This equation governs both the propagation and the reinitialization of ξ. The two parameters γ and ϵ set scales for the interface velocity and sharpness

There are occasions on which it would be convenient to extend the Level Set method to work with three or more fluids. We propose an edifice for doing so using multiple instances of the 2-fluid Level Set interface. Each interface will correspond to one of the $N \geq 3$ fluids but, as we shall see, one of the fluids will not need its own interface. Therefore, we use N-1 instances of the Level Set interface in total.

3. Example Problem

As an example, we consider an axially-symmetric example in which two droplets of fluid are suspended in a third background fluid (see Fig. 1).[5] There are $N = 3$ fluids in all, and we shall use N-1=2 copies of the Level Set interface. One of the interfaces corresponds to the fluid of droplet #1, and the other to the fluid of droplet #2. The remaining fluid does not have an interface. Each Level Set interface has an associated auxiliary function $\xi_j(\vec{r},t)$, $1 \le j \le (N-1)$, the values of which establish the boundaries of the corresponding fluid j. In the present example, there are two auxiliary functions $\xi_1(\vec{r},t)$ and $\xi_2(\vec{r},t)$ that track droplets #1 and #2, respectively.

Note that, in general, it is not the number of droplets that is relevant, but the number of distinct fluids. We could have, for example, ten droplets composed of fluid 1 and five droplets composed of fluid 2, and we would still require just two Level set interfaces.

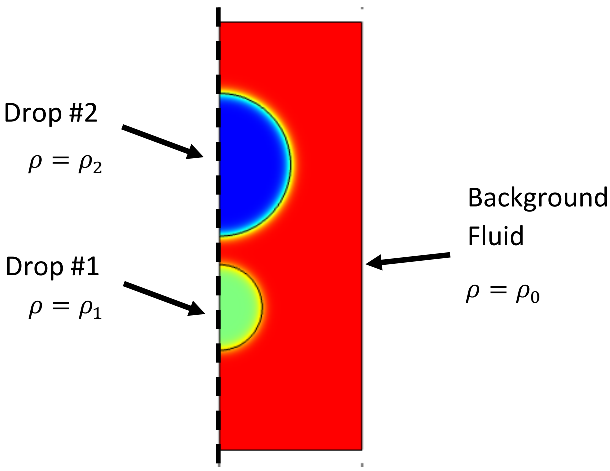

Fig. 1: Test problem involving three fluids. Two dissimilar droplets are suspended in a third fluid.

In each of the Level Set interfaces j, two fluids are modeled. The first is the fluid j tracked by the auxiliary function $\xi_j(\vec{r},t)$ of that interface. The other is a composite fluid, made up of all of the other fluids. Since we are restricting our attention to the case of laminar flow, the fluids do not mix, and we can express the density ρ and viscosity ν of the composite fluid as:

$$\rho = \rho_0 + \sum_{k \neq j} (\rho_k - \rho_0) \cdot (\xi_k(\vec{r},t) > 0.5)$$

$$\nu = \nu_0 + \sum_{k \neq j} (\nu_k - \nu_0) \cdot (\xi_k(\vec{r},t) > 0.5)$$

In this way, the fluid properties necessary to mutually solve each of the N-1 Level Set problems become fully defined.

More specifically, in the example problem with three fluids, we have a background fluid with density ρ_0 and viscosity ν_0, and we have N-1=2 Level Set interfaces. The first interface models droplet #1 with properties ρ_1, ν_1 that is suspended in a composite fluid having:

$$\rho = \rho_0 + (\rho_2 - \rho_0) \cdot (\xi_2(\vec{r},t) > 0.5)$$

$$\nu = \nu_0 + (\nu_2 - \nu_0) \cdot (\xi_2(\vec{r},t) > 0.5)$$

The second interface models droplet #2 with properties ρ_2, ν_2 that is suspended in a composite fluid with:

$$\rho = \rho_0 + (\rho_1 - \rho_0) \cdot (\xi_1(\vec{r},t) > 0.5)$$

$$\nu = \nu_0 + (\nu_1 - \nu_0) \cdot (\xi_1(\vec{r},t) > 0.5)$$

It is the auxiliary functions $\xi_1(\vec{r},t)$ and $\xi_2(\vec{r},t)$ that couple the two interfaces.

Fig. 2 shows an example solution in which the system was begun in the configuration of Fig. 1. All three fluids have a viscosity of 100 mPa-s. The background fluid has a density of $\rho_0 = 1$ g/cm^3, the lower fluid drop a density ($\rho_1 = 0.6$ g/cm^3) and the upper fluid drop a density (ρ_2=0.8 g/cm^3). In the course of the simulation, the lower drop rises more quickly than the upper drop, and overtakes it. The upper drop then wraps around the lower drop as shown.

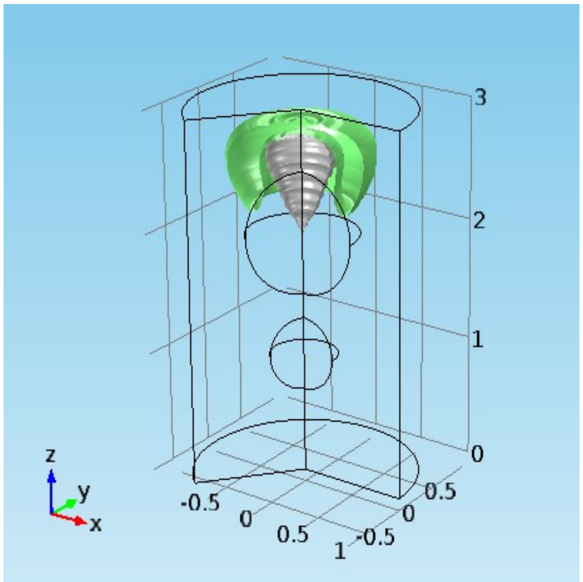

Fig. 2: Solution at t=0.3 sec for a Level Set test problem with three fluids

4. Dilute Species Method

An interesting aspect of this simulation is that each auxiliary function "travels with" one of the fluids and maintains the knowledge of where its boundaries are. There are, in fact, other means of carrying out this role without using a Level Set interface.

One might, for example, use the analogy of a chemical tracer incorporated into each fluid type. As long as the flow is laminar, the fluids do not mix. Therefore, if the diffusion constant is made sufficiently small, the tracers will not travel from fluid-to-fluid. In that case, the tracers travel with their respective fluids and can be used to detect which fluid type exists at each physical location.

In this case, the model is built around a **Transport of Dilute Species** interface with N-1 species having concentrations c_j.

The geometry defines regions for the starting disposition of each of the N fluid types as before, and for N-1 of the fluid types, we assign one of the dilute species as a tracer. The tracers are initialized with unit concentration $c_j = 1\left[\frac{M}{m^3}\right]$ in regions initially composed of the j^{th} fluid type and $c_j = 0\left[\frac{M}{m^3}\right]$ in all other regions. One of the fluid types (j=0) is distinguished by the fact that all of the tracer concentrations are zero.

The next step is to implement a Navier Stokes Laminar Flow interface for a single fluid, the properties of which depend on the local concentration of dilute species.

$$\rho(\vec{r},t) = \rho_0 + \sum_{k=1}^{N-1}(\rho_k - \rho_0)\cdot\left(c_k > 0.5\left[\frac{M}{m^3}\right]\right)$$

$$\nu(\vec{r},t) = \nu_0 + \sum_{k=1}^{N-1}(\nu_k - \nu_0)\cdot\left(c_k > 0.5\left[\frac{M}{m^3}\right]\right)$$

The solution then proceeds as it did when using the Level Set interfaces. Fig. 3 shows the result of the same test problem that was shown in Fig. 2, but this time solved by using the "Dilute Species" method. The result is clearly quite similar.

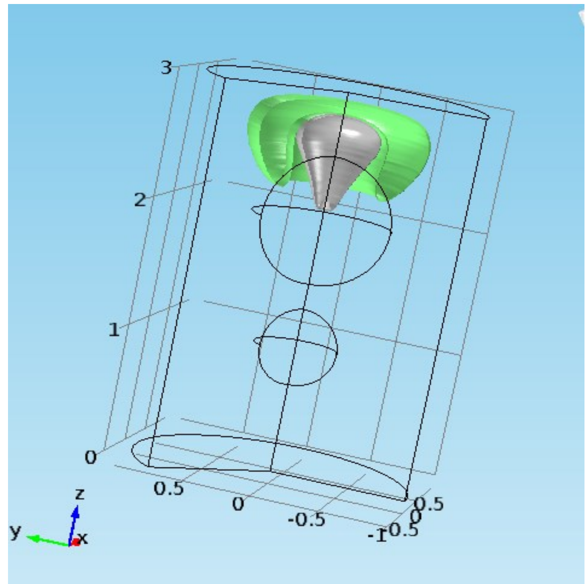

Fig. 3: Solution at t=0.3 sec of a Dilute Species version of the same problem shown in Fig. 2.

The principle reason for using COMSOL Level Set interfaces rather than this alternate scheme is that the Level Set interface implements surface tension. It is computed from the auxiliary function ξ by using $p = \gamma\nabla\cdot\vec{n}$, where γ is the surface tension and $\vec{n}$ is the surface normal $\vec{n} = \nabla\xi/|\nabla\xi|$. Smoothed versions of the dilute species concentrations c_k could, in principle, also be used to formulate an expression for the surface tension, but to date a procedure has not been formulated and tested. Thus, we currently use the Dilute Species formulation for qualitative work, or when we believe the surface tension to be a secondary consideration.

5. Conclusions

We have demonstrated two methods of solving laminar multiphase fluid-dynamic problems with more than two fluids. One builds on the capabilities of the COMSOL Level Set interface, and the other uses the COMSOL Transport of Dilute Species interface. The Dilute Species method is quick to set up, and has been useful to check the Level Set solution when the surface tension is zero. However, the use of multiple level-set interfaces is recommended when the surface tension is an important consideration.

6. References

1. Sethian, J. A., Level Set Methods and Fast Marching Methods: Evolving Interfaces in Computational Geometry, Computer Vision and Materials Science, *Cambridge Monographs on Applied and Computational Mathematics*, Cambridge University Press, New York, NY (1999).

2. Chunming Li et al., Level Set Evolution without Re-Initialization: A New Variational Formulation, *Proc. Of the 2005 IEEE Computer Society Conf. on Computer Vision and Pattern Recognition*, **Vol. 1**, pp. 430-436 (2005).

3. Chunming Li et al., Distance Regularized Level Set Evolution and its Application to Image Segmentation, *IEEE Trans. On Image Processing*, **Vol 19**, No. 12, pp. 3243-3254 (2010).

4. J. McCaslin and O. Desjardins, A localized re-initialization equation for the conservative level set method, *Journal of Computational Physics* 262, pp. 408-426 (2014).

5. The example problem is inspired by a previous example **“rising bubble 2daxi”,** which may be found in the “CFD Module/Multiphase Tutorials” section of the COMSOL Multiphysics Application Library.