A Consistent Rescaled Momentum Transport Method for Simulating Large Density Ratio Incompressible Multiphase Flows using Level Set Methods

S. Ghods and M. Herrmann
School for Engineering of Matter, Transport and Energy
Arizona State University
Tempe, AZ 85287

Abstract. Many multiphase flows relevant to natural phenomena and technical applications are characterized by large density ratios between the phases or fluids. Numerical simulations of such flows are especially challenging if the phase interface has a complex shape and is subject to large shear. This scenario is typical for atomization of liquids under ambient conditions. In this paper we discuss some of the reasons why one-fluid level-set based methods are prone to become unstable for high-density ratio/high shear atomizing flows and present a consistent rescaled momentum transport (CRMT) method that addresses the identified shortcomings. We present results obtained with the new method for a number of high-density ratio test cases, including the advection of a 1,000,000:1 density ratio impulsively accelerated drop, a 1000:1 density ratio damped surface wave, and the collapse of a water column in air under ambient conditions. The new method shows significantly improved results compared to standard level set based single-fluid methods.
Introduction

Many multiphase flows occur at ambient conditions resulting in large density ratios between the fluids and involve significant shear at the phase interface. A prime example for such a scenario involves the atomization of liquids. Although many atomization devices for technical applications under real operating conditions operate in pressurized gaseous environments, such devices are often studied experimentally under ambient pressure conditions to lower experimental cost. As such, significant more experimental data exists for ambient, i.e. large density ratio conditions than for high pressure, i.e. low density ratio conditions. Experimental datasets necessary for validation of simulation tools are thus more readily available and often of higher fidelity for high density ratio than for low density ratio applications.

The case is reversed for detailed numerical simulations of atomizing flows. There, a large density ratio can result in a more stiff system of equations that are more costly to solve. Unfortunately, some classes of numerical techniques describing the dynamics of the phase interface in incompressible flows, mostly level set based methods, are prone to numerical instabilities if the liquid-to-gas density ratio is large, i.e. of the order of 100 or more, and the flow exhibits a significant shear at the phase interface, common to many atomization devices. This numerical instability manifests itself in a sudden spike in local velocity that can grow unbounded. A good review of past work on different numerical methods to overcome this issue can be found in [16].

Interestingly enough, some numerical methods to describe the phase interface motion appear not to be susceptible to the numerical instability, among them the geometric transport VOF methods, see [19] and references therein. The important difference of these methods to most level set based approaches lies not only in the fact that the VOF approaches solve the momentum equation in conservative form, but more importantly that they employ discrete operators for the convection terms in the momentum and VOF-scalar equation that are discretely identical. They thus ensure that mass, in the form of the VOF scalar, and momentum are transported in exactly the same discrete manner. For example Rudman [17] introduced a method to solve multiphase flows with high density ratio using a projection method on staggered grids with mass conservation based on a volume tracking method using a grid twice as fine as the velocity-pressure grid. Consistent mass and momentum advection was achieved by solving the conservative form of the momentum advection term with mass flux densities obtained from the volume-of-fluid-method. Bussmann et al. [1] extended Rudman’s algorithm to three-dimensional unstructured flow solvers, using a collocated arrangement of variables on a single mesh.

Standard level set based methods, on the other hand, transport mass and momentum in entirely different ways. Mass is transported by solving the level set equation, whereas momentum is transported using typically a non-conservative formulation of the Navier-Stokes equations. Thus, even a small error in the position of the phase interface can lead to strong generation of artificial momentum in the presence
A CRMT Method for Large Density Ratio Incompressible Multiphase Flows

of large density ratios and large shear, see Fig. 1. A literature survey shows that different numerical methods have been suggested to overcome this issue.

![Figure 1: Generation of artificial momentum when solving the momentum equation in non-conservative form.](image)

Smiljanovski et al. [20] proposed an in-cell reconstruction hybrid tracking/capturing method for deflagration discontinuities. With it, the front geometry is explicitly computed using a level set scalar. This is then used to reconstruct consistent values in both fluids using the known jump conditions across the interface and consistent cell face fluxes using only values in the respective fluids. The method was successfully applied to density discontinuities in the context of Richmyer-Meshkov instabilities [5].

Hu et al. [7] constructed a method based on a standard Cartesian finite volume method and the level set technique for multi-fluid problems with complex boundaries. Two sets of equations are solved for each fluid, and an interface interaction method [6] is used to exchange momentum between the two fluids. Cell-face apertures are calculated according to the level set distribution along the Cartesian grid cell faces, in order to determine the amount of momentum transferred along the interface.

To avoid the artificial generation of momentum depicted in Fig. 1 one could of course switch from the non-conservative form of the Navier-Stokes equations to the conservative form and use a discretely conservative numerical scheme. However, this approach is equally bound to fail, since the density necessary to reconstruct velocity from momentum is again prone to phase interface position errors resulting in large overshoots in velocity. Minimizing position errors of the phase interface can alleviate the problem, however, even if a level set method were mass conserving, there still exists the mechanism of artificial momentum/velocity creation since momentum and mass are not guaranteed to be transported in a discretely consistent manner.

The key in avoiding the numerical instability is thus to ensure a discretely consistent
transport of mass and momentum. To this end, Sussman et al. [22] suggested a method that uses the coupled level set and volume-of-fluid method [21], where they extend the velocity of the fluid with the higher density to the fluid with the lower density in a band around the fluid interface and store two different velocity fields. Transport of level set and volume-of-fluid functions are done using the extrapolated (higher density fluid) velocity.

Raessi [13] and Raessi & Pitsch [16] introduced a method to construct flux densities [18] from level set scalar information and use these flux densities to transport momentum in a consistent manner. However, their method is presently limited to one- and two-dimensional cases and not straightforward to extend to three dimensions.

In this paper we introduce an alternative approach to ensure discretely consistent transport of mass and momentum for level set based methods that is simple to implement, viable in three dimensions, applicable to unstructured, collocated finite volume formulations of the governing equations and consistent with the balanced force formulation for level set methods [3].

**Governing equations**

The equations governing the motion of an unsteady, incompressible, immiscible, two-fluid system are the Navier-Stokes equations in conservative form,

\[
\frac{\partial \rho \mathbf{u}}{\partial t} + \nabla \cdot (\rho \mathbf{u} \mathbf{u}) = -\nabla p + \nabla \cdot \left( \mu \left( \nabla \mathbf{u} + \nabla^T \mathbf{u} \right) \right) + \rho \mathbf{g} + \mathbf{T}_\sigma
\]  

(1)

or in non-conservative form,

\[
\frac{\partial \mathbf{u}}{\partial t} + \mathbf{u} \cdot \nabla \mathbf{u} = -\frac{1}{\rho} \nabla p + \frac{1}{\rho} \nabla \cdot \left( \mu \left( \nabla \mathbf{u} + \nabla^T \mathbf{u} \right) \right) + \mathbf{g} + \frac{1}{\rho} \mathbf{T}_\sigma
\]  

(2)

where \( \mathbf{u} \) is the velocity, \( \rho \) the density, \( p \) the pressure, \( \mu \) the dynamic viscosity, \( \mathbf{g} \) the gravitational acceleration, and \( \mathbf{T}_\sigma \) the surface tension force which is non-zero only at the location of the phase interface \( \mathbf{x}_f \),

\[
\mathbf{T}_\sigma(\mathbf{x}) = \sigma \kappa \delta(\mathbf{x} - \mathbf{x}_f) \mathbf{n}
\]  

(3)

with \( \sigma \) the assumed constant surface tension coefficient, \( \kappa \) the local mean surface curvature, \( \mathbf{n} \) the local surface normal, and \( \delta \) the delta-function. In addition, conservation of mass results in the continuity equation,

\[
\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \mathbf{u}) = 0.
\]  

(4)

The phase interface location \( \mathbf{x}_f \) between the two fluids is described by a level set scalar \( G \), with

\[
G(\mathbf{x}_f, t) = 0
\]  

(5)

at the interface, \( G(\mathbf{x}, t) > 0 \) in fluid 1, and \( G(\mathbf{x}, t) < 0 \) in fluid 2. Differentiating (5) with respect to time yields the level set equation,

\[
\frac{\partial G}{\partial t} + \mathbf{u} \cdot \nabla G = 0
\]  

(6)
A CRMT Method for Large Density Ratio Incompressible Multiphase Flows

For numerical accuracy of geometric properties of the phase interface, it is advantageous, although not necessary, to define the level set scalar away from the interface to be a signed distance function,

$$|\nabla G| = 1$$ (7)

Assuming $\rho$ and $\mu$ constant within each fluid, density and viscosity at any point $x$ can be calculated from

$$\rho(x) = H(G)\rho_1 + (1 - H(G))\rho_2$$ (8)

$$\mu(x) = H(G)\mu_1 + (1 - H(G))\mu_2$$ (9)

where indices 1 and 2 denote values in fluid 1, respectively 2, and $H$ is the Heaviside function. Finally, the interface normal vector $n$ and the interface curvature $\kappa$ can be expressed in terms of the level set scalar as

$$n = \frac{\nabla G}{|\nabla G|}, \quad \kappa = \nabla \cdot n.$$ (10)

Numerical Method

The Navier-Stokes equations are solved using a fractional step method [8] on unstructured collocated meshes using a finite volume approach, where control volume material properties like density and viscosity are defined using Eqs. (8) and (9) as

$$\rho_{cv} = \psi_{cv}\rho_1 + (1 - \psi_{cv})\rho_2$$ (11)

$$\mu_{cv} = \psi_{cv}\mu_1 + (1 - \psi_{cv})\mu_2,$$ (12)

with the control volume volume fraction $\psi_{cv}$ given by

$$\psi_{cv} = \frac{1}{V_{cv}} \int_{V_{cv}} H(G) dV.$$ (13)

Here $V_{cv}$ is the volume of the control volume.

In the Consistent Rescaled Momentum Transport (CRMT) method, we first solve the continuity equation, Eq. (4). In discrete form this results in

$$V_{cv} \frac{\rho_{cv}^n - \rho_{cv}^{n-1}}{\Delta t} + \sum_f u_f^n \rho_f^A_f = 0,$$ (14)

where $A_f$ is the cell face area, $u_f^n$ is the face normal velocity, and $\rho_f^n$ is calculated from Eqs. (11) and (13) using the level set solution at time $t^n$. In Eq. (14), $\rho_f^A$ is defined as

$$\rho_f^A = \begin{cases} \rho_{upwind}^n & \epsilon < \psi_{cv} < 1 - \epsilon \quad \text{or} \quad \epsilon < \psi_{nbr} < 1 - \epsilon \\ \frac{\rho_{cv}^n + \rho_{nbr}^n}{2} & \text{elsewhere} \end{cases}.$$ (15)

Here $\epsilon$ is a small number, the index $nbr$ denotes the neighbor control volume to $cv$ sharing the same face, and $\rho_{upwind}$ is calculated using a simple first-order upwind approach,

$$\rho_{upwind}^n = \begin{cases} \rho_{cv}^n & u_f \geq 0 \\ \rho_{nbr}^n & u_f < 0. \end{cases}$$ (16)
A CRMT Method for Large Density Ratio Incompressible Multiphase Flows

The reason for using a simple first-order approach here lies in the fact that this will guarantee boundedness of $\rho_{cv}^*$, provided the face normal velocities $u_{n}^f$ are discretely divergence free. The choice of $\rho_f^*$ away from the phase interface in Eq. (15) is dictated by the discrete operator for the momentum equation away from the interface described below and does not result in an unconditionally unstable method for Eq. (14), since away from the phase interface $\rho_{cv}^n = \rho_{nbr}^n$.

To conserve momentum discretely, we choose the conservative form of the Navier-Stokes equations. The discrete form of the conservative Navier-Stokes equations on collocated unstructured meshes reads

$$V_{cv} \frac{(\rho u)^*_{i,cv}}{\Delta t} + \sum_f u^n_f (\rho u)_{i,f}^n A_f = \sum_f A_f \mu_f ((\nabla u)_{i,f} + (\nabla u)^T_{i,f})$$

$$+ V_{cv} \rho_{cv}^n g_i + V_{cv} \rho_{cv}^n F^n_{i,cv}, \quad (17)$$

where $F^n_{i,cv}$ is a surface tension force induced acceleration, the subindex $i$ indicates a spatial direction, $\mu_f = (\mu_{cv} + \mu_{nbr})/2$, and $u_{n}^f = \rho_{cv}^n u_{n}^f$.

To ensure discrete consistency with Eq. (14), $(\rho u)_{i,f}^n$ in Eq. (17) is defined as

$$(\rho u)_{i,f}^n = \begin{cases} (\rho u)_{i,f}^n_{upwind} & \epsilon < \psi_{cv} < 1 - \epsilon \text{ or } \epsilon < \psi_{nbr} < 1 - \epsilon \\ \frac{(\rho u)_{i,cv}^n + (\rho u)_{i,nbr}^n}{2} & \text{elsewhere} \end{cases} \quad (18)$$

with $(\rho u)_{i,f}^n_{upwind}$ calculated using the first-order upwind scheme

$$(\rho u)_{i,f}^n_{upwind} = \begin{cases} (\rho u)_{i,cv}^n & u_f \geq 0 \\ (\rho u)_{i,nbr}^n & u_f < 0 \end{cases} \quad (19)$$

This ensures that the resulting method is discretely energy conserving away from the phase interface and discretely identical to the continuity equation method.

To calculate $F^n_{i,cv}$ in Eq. (17), we follow the balanced force approach for collocated, finite volume methods [3]. At the cell face, the surface tension force is

$$T^n_{\sigma_f} = \sigma \kappa^n_f (\nabla \psi)^n_f, \quad (20)$$

resulting in

$$F^n_f = T^n_{\sigma_f}/\rho_f^*, \quad (21)$$

with $\rho_f^* = (\rho_{cv}^n + \rho_{nbr}^n)/2$. To ensure discrete consistency between the surface tension force at the control volume centroid and the pressure gradient evaluated there, $F^n_{i,cv}$ is calculated from $F^n_f$ using a face-area weighted least-squares method [10] by minimizing

$$\epsilon_{cv} = \sum_f \left( F^n_{i,cv} n_{i,f} - F^n_f \right)^2 A_f, \quad (22)$$

where $n_{i,f}$ is the face normal vector.

Using the solutions to Eqs. (17) and (14), $(\rho u)_{i,cv}^*$ and $\rho_{cv}^*$, we can then calculate the predicted velocity,

$$u_{i,cv}^* = \frac{(\rho u)_{i,cv}^*}{\rho_{cv}^*} \quad (23)$$
A CRMT Method for Large Density Ratio Incompressible Multiphase Flows

Next, we project the predicted velocity field \( \mathbf{u}^\star \) into the subspace of divergence free velocity fields, by first solving the pressure Poisson equation

\[
\nabla \cdot \left( \frac{1}{\rho^n} \nabla p \right) = \frac{1}{\Delta t} \nabla \cdot \mathbf{u}^\star
\]

which in discrete form is

\[
\sum_f \frac{1}{\rho^n_f} \frac{\partial p^n}{\partial n} A_f = \frac{1}{\Delta t} \sum_f u^n_f A_f ,
\]

with

\[
u^n_f = \frac{1}{2} (u^n_{i,cv} + u^n_{i,nbr}) n_{i,f} - \frac{1}{2} \Delta t \left( F^n_{i,cv} + F^n_{i,nbr} \right) n_{i,f} + \Delta t F^n_f .
\]

The projection, i.e. correction step is then

\[
\mathbf{u}^{n+1} = \mathbf{u}^\star - \frac{\Delta t}{\rho^n} \nabla p
\]

which for the face velocities in discrete form is

\[
u^{n+1}_f = u^n_f - \Delta t P^n_f ,
\]

with

\[
P^n_f = \frac{1}{\rho^n_f} (\nabla p^n_{cv})_f = \frac{1}{\rho^n_f} \left( \frac{p^n_{nbr} - p^n_{cv}}{|s_{cv,nbr}|} \right) .
\]

Here, \( s_{cv,nbr} \) is the vector connecting the \( cv \) and \( nbr \) control volume centroids.

To correct the control volume velocities, first the control volume centroid-based density weighted pressure gradient \( P^n_{cv} \) is calculated from the face-based density weighted gradient \( P_f \) using the same face-area weighted least-squares method employed in calculating \( F_f \), see Eq. (22),

\[
\epsilon^n_{cv} = \sum_f (P^n_{i,cv} n_{i,f} - P^n_f)^2 A_f .
\]

Then, the control volume centroid velocity is corrected as

\[
u^{n+1}_{i,cv} = u^n_{i,cv} - \Delta t P^n_{i,cv} .
\]

Finally, we discard the solution to the continuity equation, \( \rho^\star \), and reset the density at \( t^{n+1} \) using the level set solution \( G^{n+1} \) obtained from solving the level set equation (6), as

\[
\rho^{n+1}_{cv} = \psi^{n+1}_{cv} \rho_1 + (1 - \psi^{n+1}_{cv}) \rho_2
\]

This density is then used to update the momentum as

\[
(\rho \mathbf{u})^{n+1}_{i,cv} = \rho^{n+1}_{cv} u^{n+1}_{i,cv} .
\]

Results

In this section, we present results obtained with the new method for a range of test cases involving large fluid to gas density ratios. In all cases, we track the position of the phase interface using the Refined Level Set Grid method [3].
A CRMT Method for Large Density Ratio Incompressible Multiphase Flows

Collapse of a Water Column

A 2D water column with initial height and width of $h=5.715$ cm is placed inside a container of size $40 \times 10$ cm as shown in Fig. 2. The density of the water and air are $\rho_l = 1000$ kg/m$^3$, respective $\rho_g = 1.226$ kg/m$^3$, the viscosities are $\mu_l = 1.137 \times 10^{-3}$ kg/ms, respective $\mu_g = 1.78 \times 10^{-5}$ kg/ms, the surface tension coefficient is $\sigma = 0.0728$ N/m, and the gravitational acceleration is $g = -9.81$ m/$s^2$.

Figure 2 shows the phase interface shape at $\Delta t = 0.1$ s, time intervals obtained using the CRMT method compared to the shape obtained using a standard non-conservative method [3]. The non-conservative method shows significantly slower lateral spread of the water column and some unphysical deformations of the phase interface as compared to the results of the CRMT method. The CRMT results are comparable to the improvements reported using the method of Raessi [14] and Raessi & Pitsch [15].

![Figure 2: Qualitative comparison of dam-break results with density ratio 815, using non-conservative method (left) and CRMT method (right) with no-slip boundary condition on the horizontal wall.](image)

The non-dimensional front position and non-dimensional height of the water column as a function of non-dimensional time are shown in Fig. 3, where reference length is $h$ and the reference time is $\sqrt{\frac{h}{g}}$. Figure 3 also shows the results of a grid refinement study resolving the container by $512 \times 128$, $1024 \times 256$, and $2048 \times 512$ equi-sized hexahedral control volumes. The lateral front position converges under grid refinement, however the results appear to converge to a slightly faster spread rate as that observed.
Figure 3: Non-dimensional front position (top) and non-dimensional height of water column (bottom) of dam-break versus non-dimensional time compared to the experimental results [11].

experimentally [11], whereas the height of the water column is well captured even on the coarsest mesh.

Table 1 shows both the order of convergence for the relative error of the front position ($Z$) with respect to the experimental data, $E_{\text{exp}} = \frac{Z_{\text{exp}} - Z}{Z_{\text{exp}}}$ and with respect to the finest grid solution, $E_{\text{finest}} = \frac{Z_{\text{finest}} - Z}{Z_{\text{finest}}}$ at time $t = 3.9$. 
A CRMT Method for Large Density Ratio Incompressible Multiphase Flows

<table>
<thead>
<tr>
<th>Grid</th>
<th>$E_{exp}$</th>
<th>order</th>
<th>$E_{finest}$</th>
<th>order</th>
</tr>
</thead>
<tbody>
<tr>
<td>128X32</td>
<td>0.356</td>
<td>-</td>
<td>0.360</td>
<td>-</td>
</tr>
<tr>
<td>256X64</td>
<td>0.190</td>
<td>0.90</td>
<td>0.195</td>
<td>0.88</td>
</tr>
<tr>
<td>512X128</td>
<td>0.058</td>
<td>1.71</td>
<td>0.064</td>
<td>1.60</td>
</tr>
<tr>
<td>1024X256</td>
<td>0.015</td>
<td>1.95</td>
<td>0.021</td>
<td>1.60</td>
</tr>
<tr>
<td>2048X512</td>
<td>0.006</td>
<td>1.32</td>
<td>-</td>
<td>-</td>
</tr>
</tbody>
</table>

Table 1: Error of the front position in the collapse of the water column.

Damped surface wave

The dynamics of a small amplitude damped surface wave between two superposed immiscible fluids are described by the initial value theory of Prosperetti [12]. The initial surface position inside a $[0, 2\pi] \times [0, 2\pi]$ box is given by a sinusoidal disturbance of wavelength $\lambda = 2\pi$ and amplitude $A_0 = 0.01\lambda$,

$$G(x, t = 0) = y - y_0 + A_0 \cos(x - h_G/2),$$

with $y_0 = \pi$. Periodic boundary conditions are used in the $x$-direction and slip walls are imposed in the $y$-direction. The initial value solution for two fluids with equal kinematic viscosity $\nu$ and $\lambda = 2\pi$ can be written as [12]

$$A_{ex}(t) = \frac{4(1 - 4\beta)\nu^2}{8(1 - 4\beta)\nu^2 + \omega_0^2} A_0 \text{erfc}\sqrt{\nu t} + \sum_{i=1}^{4} \frac{z_i}{Z_i} \left( \frac{\omega_0^2 A_0}{z_i^2 - \nu} \right) \exp[(z_i^2 - \nu)t] \text{erfc}(z_i\sqrt{t}),$$

where $z_i$ are the roots of

$$z^4 - 4\beta\sqrt{\nu}z^2 + 2(1 - 6\beta)\nu z^2 + 4(1 - 3\beta)\nu^{3/2} z + (1 - 4\beta)\nu^2 + \omega_0^2 = 0,$$

the dimensionless parameter $\beta$ is given by $\beta = \rho_1\rho_2/(\rho_1 + \rho_2)^2$, the inviscid oscillation frequency is $\omega_0 = \sqrt{\frac{\sigma}{\rho_1 + \rho_2}}$, and $Z_i = \prod_{j=1}^{4} (z_j - z_i)$. In [3] results for $\rho_l = 1000$, $\rho_g = 1$, $\sigma = 2$ and $\nu_l = \nu_g = 0.064720863$ were reported using the non-conservative formulation.

Figure 4 shows the temporal evolution of the non-dimensional disturbance amplitude $A$ for a mesh consisting of 128x128 equi-sized hexahedra using both the CRMT and non-conservative method [3], including a zoom of the temporal evolution starting at $t = 130$ to more clearly see the difference in results. The CRMT method shows noticeably improved results compared to the non-conservative methods.

Figure 5 shows the results of a grid refinement study in a zoom of the temporal evolution starting at $t = 130$. Excellent agreement of the CRMT method with the analytical results can be seen. Figure 6 shows the evolution of the corresponding non-dimensional error $E(t) = (A(t) - A_{ex}(t))/A_0$ for hexahedral and prism meshes, while Tab. 2 summarizes their root mean squares. At the same grid resolution, the CRMT method shows a significantly lower error as compared to the non-conservative formulation [3].
Figure 4: Normalized amplitude \((A/\lambda)\) of damped surface wave with density ratio 1000 versus time.

<table>
<thead>
<tr>
<th>Mesh Type</th>
<th>CRMT</th>
<th>Ref [3]</th>
</tr>
</thead>
<tbody>
<tr>
<td>Cartesian</td>
<td></td>
<td></td>
</tr>
<tr>
<td>32</td>
<td>1.440E-002</td>
<td>4.82E-002</td>
</tr>
<tr>
<td>64</td>
<td>4.534E-003</td>
<td>2.08E-002</td>
</tr>
<tr>
<td>128</td>
<td>2.465E-003</td>
<td>1.27E-002</td>
</tr>
<tr>
<td>256</td>
<td>2.246E-003</td>
<td>1.18E-002</td>
</tr>
<tr>
<td>Prism</td>
<td></td>
<td></td>
</tr>
<tr>
<td>32</td>
<td>1.263E-002</td>
<td>5.64E-002</td>
</tr>
<tr>
<td>64</td>
<td>6.743E-003</td>
<td>1.41E-002</td>
</tr>
<tr>
<td>128</td>
<td>6.188E-003</td>
<td>1.13E-002</td>
</tr>
<tr>
<td>256</td>
<td>4.703E-003</td>
<td>1.57E-002</td>
</tr>
</tbody>
</table>

Table 2: Rms of amplitude error for damped surface wave.
A CRMT Method for Large Density Ratio Incompressible Multiphase Flows

Figure 5: Normalized amplitude \((A/\lambda)\) of damped surface wave with density ratio 1000 versus time using CRMT method.

Zero gravity column oscillation

To further verify the implementation of the new method, this section presents results for zero gravity oscillating columns. The theoretical oscillation period for columns in the linear regime is given by [9]

\[
\omega^2 = \frac{n(n^2 - 1)\sigma}{(\rho_1 + \rho_2)R_0^2}.
\] (37)

In all simulations, a column of radius \(R_0 = 2\) is placed in the center of a \([-10, 10]\) square box with periodic boundary conditions on all sides and \(\sigma = 1\), \(\rho_1 = 1\), \(\rho_2 = 0.01\), \(\mu_1 = 0.01\), and \(\mu_2 = 1 \times 10^{-4}\), resulting in a Laplace number of \(\text{La} = 20000\). The column is initially perturbed by a mode \(n = 2\) perturbation with an initial amplitude of \(A_0 = 0.01R_0\). The time step size in all simulations is chosen as \(\Delta t = 0.5\Delta t_{cap}\). Where

\[
t_{cap} = \sqrt{\frac{(\rho_1 + \rho_2)h^3}{4\pi\sigma}}.
\] (38)

Table 3 shows the period of oscillation error \(E_T = |T_{calc}\omega/2\pi - 1|\) for the oscillating column together with the results reported in [3]. On fine hexahedral meshes, the CRMT method gives noticeably improved results as compared to those of the non-conservative method, whereas the results on prism meshes are comparable.

Convection of high density droplet

In this test case initially proposed by Bussman et al. [1], a 2D liquid droplet of diameter \(D = 0.4\) is placed in the center of a \(1 \times 1\) periodic domain filled with gas. The density ratio is chosen as \(\rho_l/\rho_g = 10^6\) and the fluids are assumed inviscid and without surface
Figure 6: Amplitude error E of damped surface wave for hexahedral (top) and prism (bottom) meshes using CRMT method.

Table 3: Zero gravity 2D column oscillation. Error in oscillation period as compared to linear theory [9].

<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>20/64</td>
<td>7.47e-2</td>
<td>4.04e-2</td>
<td>5.66e-2</td>
<td>5.91e-2</td>
</tr>
<tr>
<td>20/128</td>
<td>7.32e-3</td>
<td>1.05e-2</td>
<td>1.61e-2</td>
<td>1.65e-2</td>
</tr>
<tr>
<td>20/256</td>
<td>3.44e-4</td>
<td>3.7e-3</td>
<td>1.35e-2</td>
<td>1.36e-2</td>
</tr>
</tbody>
</table>

tension. The drop is given an initial homogeneous velocity of $\mathbf{u} = (1, 0)$ while the gas is initially at rest. We have employed different structured equi-sized hexahedral
A CRMT Method for Large Density Ratio Incompressible Multiphase Flows

and unstructured prism grids. Because of the large density ratio, the drop should stay essentially undeformed while passing through the computational domain multiple times.

Figure 7: Droplet shape after one passthrough using non-conservative method [3].

Figure 7 shows the drop shape after passing the domain once ($t \approx 1$) using the non-conservative method [3] on a 128x128 hexahedral mesh. Erroneous transfer of momentum from the liquid to the gas has caused significant interface deformation, resulting in an unphysical shattering of the drop. Figure 8 shows the drop shape after passing the entire domain once ($t = 1$) obtained using the CRMT method together with the expected solution for varying mesh resolutions and Tab. 4 summarizes the corresponding shape errors. While there are some minor deformations of the drop visible, the drop stays nearly spherical and no erroneous large scale interface deformations are visible.

<table>
<thead>
<tr>
<th>grid size</th>
<th>$E_T$ (hexahedra)</th>
<th>order</th>
<th>$E_T$ (prism)</th>
<th>order</th>
</tr>
</thead>
<tbody>
<tr>
<td>1/32</td>
<td>5.33e-3</td>
<td>-</td>
<td>5.42e-3</td>
<td>-</td>
</tr>
<tr>
<td>1/64</td>
<td>3.73e-3</td>
<td>0.51</td>
<td>2.93e-3</td>
<td>0.88</td>
</tr>
<tr>
<td>1/128</td>
<td>2.64e-3</td>
<td>0.49</td>
<td>1.65e-3</td>
<td>0.82</td>
</tr>
<tr>
<td>1/256</td>
<td>1.85e-3</td>
<td>0.51</td>
<td>-</td>
<td>-</td>
</tr>
</tbody>
</table>

Table 4: Shape errors for convection of high density droplet.

Summary and conclusion

A new consistent rescaled momentum transport (CRMT) scheme for modeling incompressible, multiphase flows with high density ratios in the context of level set interface capturing methods has been presented. In this method, the conservative
form of the Navier-Stokes equations is solved using an unstructured, collocated, incompressible, fractional step flow solver. Instead of replacing the continuity equation in its entirety by a level set equation to keep track of the phase interface separating fluids of different, but constant, density, we solve the continuity equation to obtain a predicted density using operators that are discretely consistent with those used in the solution of the conservative momentum equation. Using the predicted momentum and density, we recover a predicted velocity that is then projected into the subspace of divergence free velocity fields. To avoid undue dissipative errors from solving the continuity equation directly, we then reset the density and momentum according to the density obtained from a level set solution.

It should be pointed out that this new approach is neither mass nor momentum
A CRMT Method for Large Density Ratio Incompressible Multiphase Flows

conserving. The key idea is in fact to allow mass and momentum errors, where the former are unavoidable in a pure level set method, but to ensure that they are discretely consistent.

The new method has shown excellent results for a range of test cases involving large density ratios up to $10^6$ and offers a path to simulate atomizing flows with large density ratio fluids.

Acknowledgments

This work was supported by NSF grant number CBET-0853627. The authors would also like to thank M. Raessi, O. Desjardins, and F. Ham for many helpful discussions.

References

A CRMT Method for Large Density Ratio Incompressible Multiphase Flows


