A Robust All-Mach Multiphase Flow Algorithm for High-Fidelity Simulations of Compressible Atomization

M.B. Kuhn* and O. Desjardins
Sibley School of Mechanical and Aerospace Engineering
Cornell University
Ithaca, NY 14853 USA

Abstract
High-fidelity simulations can accelerate understanding and illuminate important aspects of liquid atomization in highly compressible environments. For example, simulations can provide invaluable insights on the physics of scramjet engine cold-start, thereby helping design successful injection strategies. For simulating compressible liquid-gas flows with topology changes, we have developed an all-Mach, compressible multiphase flow solver that utilizes a low dissipation transport scheme to accurately represent turbulence in smooth, single-phase regions and applies a robust semi-Lagrangian scheme to handle discontinuous transport at interfaces and shocks. We employ a pressure projection scheme to avoid acoustic limitations on the time-step size. Within this framework, we focus on specific treatments of the volume fraction, pressure, and energy equations to improve stability.

*Corresponding Author: mbk225@cornell.edu
Introduction

Simulating liquid atomization in compressible environments provides unique challenges, as flow quantities can become highly discontinuous due to shocks and phase interfaces. However, this problem is of high relevance to the development of fuel delivery systems for scramjet engines, for instance. The fuel injection strategy heavily influences the atomization process, which contributes to the evaporation characteristics and spatial distribution of the resulting spray, having consequences on the performance of the engine [1]. Improving numerical capabilities to solve relevant flows, like a liquid jet in supersonic crossflow, can add significant physical insight and introduce design principles in a context where both experimental and computational studies have been limited.

Numerical techniques capable of simulating compressible atomization are sparse in the literature, in comparison with incompressible atomization. Shukla et al. [2] combine a diffuse representation of the interface with a density-sharpening technique to significantly improve the results of shock interface interaction problems, though the sharpening technique is non-conservative. Garrick et al. [3] improve upon Shukla’s method, employing a modification of the THINC (Tangent of Hyperbola for INterface Capturing) scheme from [4] and [5]. This strategy is conservative, and they apply it to the simulation of a liquid jet in supersonic crossflow.

Diffuse interface schemes are confined to spreading the interface over a handful of cells, which has the tendency to practically reduce resolution at the interface. Jemison et al. [6] treats the phase interface sharply, representing it with a planar reconstruction within a single cell. The equations governing density and energy are solved separately in each phase, while there is a single momentum equation and a single pressure equation. Their scheme combines a directionally split semi-Lagrangian transport algorithm with a pressure projection scheme adapted from [7]. Finally, they show this method to be capable of simulating a liquid jet in subsonic crossflow.

We improve upon the framework of [6], utilizing an unsplit semi-Lagrangian transport scheme [8] along with a low-dissipation scheme adapted from [9] to better simulate turbulence. Moreover, we discuss techniques for handling multiphase cells that we have found to be critical for stability and accuracy in difficult problems like a liquid jet in supersonic crossflow.

Numerical Framework

We solve the compressible Navier-Stokes equations using a finite volume framework. Equations are written in each phase for $\alpha$, $\rho_\alpha$, $\rho_\alpha E$ (i.e., volume, mass, and energy of each phase — the $i$ subscript below represents each phase), and an additional equation for mixture momentum $\rho u$ completes the system. It is fully given as

$$\frac{\partial \alpha_i}{\partial t} + \nabla \cdot (\alpha_i u) = \alpha_i \nabla \cdot u,$$

$$\frac{\partial \rho_i}{\partial t} + \nabla \cdot (\rho_i u) = 0,$$

$$\frac{\partial \rho_i E_i}{\partial t} + \nabla \cdot \left( \rho_i E_i \alpha_i u + \nabla \cdot (\rho_\alpha u) \right) = \nabla \cdot (u \cdot \tau + k\nabla T) + u \cdot F \alpha_i,$$

and

$$\frac{\partial \rho u}{\partial t} + \nabla \cdot (\rho u \otimes u) + \nabla \cdot (p I) = \nabla \cdot (\tau + F),$$

where

$$\tau = \mu \left( \nabla u + \nabla u^T - \frac{2}{3} \nabla \cdot u I \right)$$

is the viscous stress tensor, the mixture density is $\rho = \alpha_1 \rho_1 + \alpha_2 \rho_2$, and the mixture energy is $\rho E = \alpha_1 \rho_1 E_1 + \alpha_2 \rho_2 E_2$. $F$ contains all body forces, including gravity and surface tension. The mixture viscosity $\mu$ is given by $\mu = \mu_1 \mu_2 / (\mu_1 \alpha_2 + \mu_2 \alpha_1)$.

We employ a fractional step approach to time integration, dividing the process of time advancement into three components. First, we calculate the convection of each variable, then we apply the viscous and forcing terms, and finally we solve for a pressure correction. Similar to [6], we use an additional fixed-point iteration to reduce oscillatory behavior due to the non-linearity of the momentum equation.

To solve for the convective terms, we use the volumetric transport strategy discussed in [8]. This scheme is conservative and formally bounded. To improve the accuracy of the transport of continuous variables, we use a linear reconstruction in each cell, limited by the minmod slope limiter for stability.

Viscous terms are discretized using second-order finite differences and calculated explicitly using sub-stepping for robustness. Surface tension is inserted as a pressure jump in the Helmholtz equation for pressure, as in [10]. Interface curvature is calculated using a quadratic least squares approach [11, 12].

We calculate the pressure by implicitly solving a Helmholtz equation, thereby eliminating the acoustic CFL restriction. This is particularly advantageous for atomization problems, where the speed of
sound in the liquid can be significantly higher than that of the gas and the associated CFL restriction would otherwise become a cost-intensive limit. The pressure equation we solve is adapted from [7], and is written as

\[ p^{n+1} - \rho^{n+1} \left(c^*\right)^2 \Delta t^2 \nabla \cdot \left( \frac{\nabla p^{n+1}}{\rho^{n+1}} \right) = p^* - \rho^{n+1} \left(c^*\right)^2 \Delta t \nabla \cdot \mathbf{u}^*. \] (6)

This pressure is then used to correct the flow variables. Only the pressure at \( n + 1 \) is employed in the pressure correction. This is consistent with the equation from [7] and agrees with the common practice of biasing the future pressure, like in [9, 13].

In applying this pressure projection approach to a multiphase problem, decisions have to be made regarding how to calculate the advected pressure (\( p^* \)) and the bulk modulus (\( \rho c^2 \)) in cells that contain more than one phase. The solution of the pressure equation applies to both phases, but the calculation of the advected pressure and the bulk modulus must come from some combination of both equations of state, using information from both phases. The methods used for multiphase cells can have significant consequences on accuracy and stability, so careful treatment is required. We discuss the handling of these terms later.

**Hybrid Framework**

Although the unsplit sem-Lagrangian transport scheme is robust, it also introduces significant unphysical kinetic energy dissipation. In [14], we discussed hybridization of our numerical solver by introducing centered schemes in regions that require less robustness. Since then, we have improved our approach by adapting concepts from [9]. We use their discretization of the momentum equation and the kinetic energy fluxes, integrated with sem-Lagrangian transport of the density and internal energy. In situations of constant density and internal energy, results are identical to those in [14], but this modification adds robustness that was lacking when solving flows with high density and internal energy variations present.

**Multiphase Treatment**

*Mixture Rules and Liquid Volume Fraction Calculation*

Although many aspects of our numerical approach maintain a sharp interface between gas quantities and liquid quantities, the pressure projection method requires there to be a single pressure field. Solving the Helmholtz equation requires a systematic approach to designating a single advected pressure and a single bulk modulus in cells that have two phases, where each phase has its own equation of state. We compare two approaches based on common practices with a new approach that we have designed. All examples consider the stiffened gas equation of state, written as

\[ p = (\gamma - 1)(\rho E - \frac{1}{2}(\rho \mathbf{u} \cdot \mathbf{u}) - \gamma p^\infty) \] (7)

Jemison et al. suggest using the equation of state of the phase occupying the most volume in the cell, that is, the majority phase. However, this simple approach is dramatically inaccurate in the compressible atomization simulations we have performed.

A different straightforward approach to calculating these quantities is to assume isobaric closure between phases, and formulate an equation of state for the mixture. This involves calculating mixture properties based on the volume fraction and the properties of each individual phase, shown below.

\[ \frac{1}{\gamma - 1} = \sum_i \frac{\alpha_i}{\gamma - 1} \] (8)

\[ p^\infty = \frac{\gamma - 1}{\gamma} \sum_i \alpha_i \frac{\gamma_i p_i^\infty}{\gamma_i - 1} \] (9)

These mixture properties can be used to calculate the advected pressure and speed of sound in the Helmholtz equation. Although they do not use these relations in the context of a pressure projection scheme, these relations are consistent with [2] and [15]. This approach more closely aligns with a diffuse interface strategy.

A third approach is derived from the definition of isentropic bulk modulus, and follows from the work of Alahyari Beig and Johnsen [15]. In the presence of nonzero dilatation, they note that a source term should be added to the volume fraction evolution equation. In describing this approach, we refer to the use of the volume fraction source term, which they derive, and corresponding mixture rules, which we put forth.

This source term takes into account the difference in compressibility between phases when more than one phase occupies a cell. In essence, dilatation should apply more to the more compressible phase than one phase occupies a cell. In essence, dilatation should apply more to the more compressible phase than one phase occupies a cell. In essence, dilatation should apply more to the more compressible phase than one phase occupies a cell. In essence, dilatation should apply more to the more compressible phase than one phase occupies a cell.
In this equation, $K$ represents the isentropic bulk modulus, so for an individual fluid, $K_i = \rho_i c_i^2$. The isentropic bulk modulus of the mixture is defined as
\[
\frac{1}{K} = \sum_i \frac{\alpha_i}{K_i}.
\]  
(12)

To calculate this term during the transport step, we

1. Use the equation of state to calculate the pressure in each phase
\[
p_i^n = EOS_i(\rho E_i, \rho_i, u)
\]
2. Primitively transport the pressure in each phase, and the volume fraction
\[
p_i^* = p_i^n - \Delta t u \cdot \nabla p_i
\]
\[
\alpha_i^* = \alpha_i^n - \Delta t u \cdot \nabla \alpha_i
\]
3. Calculate the bulk modulus of each phase using the transported pressure
\[
K_i = (\rho c_i^2)_i^* = EOS_i(p_i^*)
\]
4. Calculate the bulk modulus of the mixture
\[
K = \rho c^2 = \left[ \sum_i \frac{\alpha_i^* (\rho c_i^2)_i^*}{\rho_i} \right]^{-1}
\]
5. Update the volume fraction
\[
\alpha_i^{n+1} = \alpha_i^* + \Gamma_i \nabla \cdot u
\]

Since this term modifies the volume fraction, it indirectly modifies all of the transported variables and can significantly influence the behavior of quantities in interface cells, particularly when dilatation is substantial. This formulation of the mixture bulk modulus also points to using a different calculation of mixture terms in the pressure equation. If we state that the evolution equation for pressure of a phase $i$ is
\[
p^{n+1} = p_i^* - \Delta t (\rho c_i^2)_i \nabla \cdot u^{n+1},
\]  
(13)
and the bulk modulus of the mixture is defined as in Equation 12, then the advected pressure of each phase must be combined in a consistent way to produce the mixed equation
\[
p^{n+1} = p^* - \Delta t \rho c^2 \nabla \cdot u^{n+1}.
\]  
(14)

This relationship informs us that
\[
p^* = \rho c^2 \sum_i \frac{\alpha_i}{(\rho c_i^2)_i} p_i^*.
\]  
(15)

### Kinetic Energy Exchange

Although using this volume fraction source term and corresponding mixture rules can add robustness, there remain other issues to be addressed. Special attention must be paid to the consequences of treating each fluid separately but using a single velocity for both fluids. In the process of convection, we transport the momentum of each phase separately. After all variables have been transported, we use the following relation to calculate the velocity in the cell, which is assumed to be the same in both phases. The $*$ indicates the quantity calculated by the convective step.
\[
u^* = \frac{\sum \alpha_i (\rho u)_i^*}{\sum \alpha_i \rho_i}
\]  
(16)

This effectively modifies the kinetic energy of each phase, but the total energy variable has not been modified. Lei and Li [16] present a term that approximates the kinetic energy exchange that takes place between two phases when the velocity is “uniformized”, as they put it.

### Work Done by Pressure

The final area where problems can arise is in calculating the work done by pressure, which is the term in the total energy equation denoted as $\nabla \cdot (p \dot{u})$. To determine the proper treatment of this term, we look individually at the contributions from the internal energy equation and from the kinetic energy equation. For the sake of brevity, let us define
\[
\frac{D}{Dt} f = \frac{\partial f}{\partial t} + \nabla \cdot (f \dot{u}).
\]

The evolution of internal energy in each phase follows the equation
\[
\frac{D}{Dt} \rho_i e_i = -p_i \nabla \cdot u_i,
\]  
(17)

leading to the mixture evolution equation
\[
\frac{D}{Dt} \rho e = -\sum_i \alpha_i p_i \nabla \cdot u_i.
\]  
(19)

Since we assume that the pressure and velocity are continuous in both phases, this simplifies to
\[
\frac{D}{Dt} \rho e = -\sum_i \alpha_i p \nabla \cdot u.
\]  
(20)

The kinetic energy equation, however, is more complicated, since the velocity is shared between phases.
The kinetic energy is defined as
\[
\rho k = \sum_i \frac{1}{2} \rho_i \rho_i [u_i^2 + v_i^2 + w_i^2] = \sum_i \alpha_i \rho_i k. \tag{21}
\]

Rephrasing this into only conserved variables, the definition becomes
\[
\rho k = \sum_i \frac{1}{2} \frac{\rho_i}{\rho^2} [(\rho u_i)^2 + (\rho v_i)^2 + (\rho w_i)^2]. \tag{22}
\]

To derive the governing equation for kinetic energy in a phase, we apply the operator \(\frac{D}{Dt}\) along with the chain rule.
\[
\frac{D(\rho k)_i}{Dt} = \frac{D\rho_i}{Dt} \rho k + \frac{D\rho}{Dt} \left( -\frac{\rho_i}{\rho} 2 k \right) + \frac{D\rho u_i}{Dt} \rho \cdot u + \frac{D\rho v_i}{Dt} \rho \cdot v + \frac{D\rho w_i}{Dt} \rho \cdot w, \tag{23}
\]

The density transport terms are equal to zero, and the momentum transport terms can be replaced by their equivalent pressure gradients.
\[
\frac{D(\rho k)_i}{Dt} = -\frac{\partial}{\partial x} \rho \rho_i \frac{\partial u_i}{\partial x} - \frac{\partial}{\partial y} \rho \rho_i \frac{\partial v_i}{\partial y} - \frac{\partial}{\partial z} \rho \rho_i \frac{\partial w_i}{\partial z} \quad \tag{24}
\]

\[
\frac{D(\rho k)_i}{Dt} = -\frac{\rho_i}{\rho} \rho \cdot \nabla p \quad \tag{25}
\]

This analysis indicates that the pressure term in the total energy equation should not simply be written
\[
\frac{D(\rho E)_i}{Dt} = -\frac{\rho_i}{\rho} \nabla \cdot (p u) \quad \tag{26}
\]

or
\[
\frac{D(\rho E)_i}{Dt} = -\nabla \cdot (p u)
\]

rather, it should instead be written
\[
\frac{D(\rho E)_i}{Dt} = -p \nabla \cdot u - \frac{\rho_i}{\rho} u \cdot \nabla p \quad \tag{27}
\]

In single-phase cells, this makes no difference. However, in multiphase cells, this alters the effect that dilatation and pressure gradient have on the total energy of each phase. Discretized directly, this formulation does not conserve the total energy of the mixture, so we rephrase the expression as
\[
\frac{D(\rho E)_i}{Dt} = -p \nabla \cdot u - \frac{\rho_i}{\rho} (\nabla \cdot (p u) - p \nabla \cdot u), \quad \tag{28}
\]

which conserves the total energy of the mixture.

### Results and Discussion

In order to evaluate the accuracy and stability of these methods, we apply each method to three test cases. Table 1 shows the five approaches that we evaluate. The first three schemes compare different mixture rules, the fourth introduces the kinetic energy exchange term from [16], and the fifth uses the discretization of the work done by pressure that we have proposed in this work with \(28\). Note that schemes 1-4 use the discretization of that term from [6], shown by \(26\).

<table>
<thead>
<tr>
<th>Scheme</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Majority-phase equation of state</td>
</tr>
<tr>
<td>2</td>
<td>Mixture rules from [2] and [15]</td>
</tr>
<tr>
<td>3</td>
<td>Volume fraction source term with associated mixture rules</td>
</tr>
<tr>
<td>4</td>
<td>Scheme 3 with kinetic energy exchange</td>
</tr>
<tr>
<td>5</td>
<td>Scheme 4 with proposed discretization of the work done by pressure</td>
</tr>
</tbody>
</table>

### 1D water-air shock-interface interaction

The first test case is from [2], where a water-air interface is accelerated by a strong shock in water. This leads to a strong rarefaction wave reflected back into the water, while a weak shock is transmitted through the air. We solve this problem with a grid of 400 points and a constant time step size of 3.8 \(\mu\)s. This test case does little to distinguish between the different schemes, and it suggests that each scheme is reasonable.

Scheme 3, however, crashes on the fifth time step and cannot complete the simulation. As the liquid momentum increases the velocity of the interface cell, the pressure in the gas dramatically drops, to the point that it becomes negative. The kinetic energy exchange term included in schemes 4 and 5 is necessary for stability. Schemes 1 and 2, which lack the kinetic energy exchange term, do not crash. We suspect that this is because they lack the source term in the volume fraction equation. Without this term, the compressibility of each phase is enforced less strictly. When the rarefaction wave begins in the liquid, schemes 1 and 2 allow for more expansion of the liquid than the other schemes, which allows for changes in the neighboring gas to be less drastic, thus avoiding a crash. Despite this difference, the curves are remarkably similar by the time shown in Figure 1, which highlights the subtlety involved with deciding the optimal treatment of multiphase cells.
The density, velocity, pressure, and volume fraction for the one-dimensional water-air shock-interface interaction at a time of 1.5 ms. Schemes 1 and 2 ( and , respectively) are essentially coincident, and appear to be slightly smoother than schemes 4 and 5 ( and , respectively), but differences are slight.

**1D rarefaction-droplet interaction**

The next test case is the interaction between a rarefaction wave, originating in the gas phase, with a one-dimensional “droplet”. During the simulation, we use the stiffened gas equation of state with the parameters listed in Table 2. The length of the domain is 20 mm, and a droplet of width 0.3 mm is placed at a position of 6.7 mm. The initial liquid density is 1000 kg/m$^3$, and the initial gas density is 1.2 kg/m$^3$. The initial pressure is uniform at 101325 Pa. The velocity is initialized with a discontinuity at 7.8 mm, where the velocity is 0 to the left of the discontinuity and 224.786 m/s to the right of the discontinuity. This velocity discontinuity creates a rarefaction wave that travels toward the droplet. This wave is partially transmitted to the droplet and induces the droplet to move to the right as the gas on the left side of the droplet expands. We use a uniform grid of 401 points and a constant time step of 0.18 µs.

<table>
<thead>
<tr>
<th>Phase</th>
<th>$\gamma$</th>
<th>$p^\infty$ (Pa)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Gas</td>
<td>1.4</td>
<td>0</td>
</tr>
<tr>
<td>Liquid</td>
<td>4.4</td>
<td>$6 \times 10^8$</td>
</tr>
</tbody>
</table>

**Table 2.** Material properties used for each phase in the second and third test cases. The expression for the equation of state is shown by (9).
This test case is of interest because it showcases a wave originating in the gas phase impacting the liquid. Additionally, the response of the liquid phase causes the droplet to move, and the combination between the motion of the droplet and rarefaction wave elucidates differences between the schemes. As the droplet moves slowly through the domain, containing information from its response to the rarefaction wave, the differences between the schemes become distinguishable as artifacts in the pressure and velocity, shown in Figure 2. In addition to those quantities, significant differences arise in the liquid gas density and the liquid pressure, which are insets in Figure 2.

Using some assumptions, we can estimate the expected variation in density and pressure in the liquid. Treating the liquid-gas interface as a stationary wall with an impedance mismatch, where the acoustic impedance of each phase is equal to $\rho c$, the initial pressure variation of -38.6 kPa caused by the rarefaction wave should almost double to -77.1 kPa in the liquid. Then, assuming the transport of the wave is isentropic, the expected density variation is -0.0003 kg/m$^3$. Though these numbers rely on approximations and ignore the additional reflections that will occur as time continues, they indicate the order of
magnitude to expect from the simulation. At the time step shown in Figure 2, schemes 1 and 2 led to liquid density variations of 2 kg/m$^3$ and liquid pressure variations of 450 kPa, while schemes 3 through 5 led to liquid density variations of -0.015 kg/m$^3$ and liquid pressure variations of -50 kPa. In addition to these errors in the liquid phase, the pressure solution and velocity solution appear more oscillatory for schemes 1 and 2 than for the other schemes.

From this test case, it seems schemes 1 and 2 allow the liquid density to deviate more than it should, and they allow the pressure solution to deviate strongly from the pressure calculated by the equation of state. This behavior contributes to inaccurate oscillations in the pressure and velocity, weakening the stability of the solver.

### 2D shock-water column interaction

Finally, we look at a test case closer to the intended application, which is atomization. This test case is the breakup of a water column by a high speed flow behind a shock wave of M=1.47, from [17]. However, we use a smaller water column with about 6 cells/diameter, so that we can probe the accuracy and stability of the method in handling under-resolved liquid structures which arise during atomization processes. The square domain is 100 mm long, and the shock is located at 48.4 mm. The droplet has a diameter of 1.6 mm and is located at 50 mm. The uniform grid has 400 points in each direction, and we run the simulation at a convective CFL of 0.9. We incorporate additional physics, with a gas viscosity of $1.78 \times 10^{-5}$ Pa s, a liquid viscosity of $1.137 \times 10^{-3}$ Pa s, and a surface tension coefficient of 0.078 N/m.

As the shock impacts the droplet and atomizes it, we investigate the evolution of the liquid density and liquid pressure. To get a sense of the variation of the liquid density, we plot its maximum and minimum obtained by each scheme over the course of the simulation. Results from scheme 1 are shown in Figure 3. This scheme yields grossly unrealistic results, with the maximum liquid density increasing sevenfold while the minimum liquid density plummets to almost zero. This 2D flow reveals that scheme 1 fails to get the liquid to behave like a liquid, due to how the liquid equation of state is only considered if the liquid takes up more than half of a cell.

In Figure 4, we compare the results from the other schemes. Schemes 3 through 5 outperform scheme 2 in maintaining a near-constant liquid density as the simulation progresses. Although scheme 2 does better than scheme 1, the density still varies rapidly and unrealistically.

Next, we plot the evolution of the minimum liquid pressure, shown in Figure 5. We look at the minimum because when the pressure becomes too low, the code can crash, so this quantity is relevant to stability. The variations in pressure from scheme 2 correspond strongly with the variations in density from Figure 4. Because of the “stiffness” introduced in the liquid equation of state, small variations in liquid density can lead to huge variations in liquid pressure. Schemes 3 through 5 steadily im-

![Figure 3. Liquid density bounds during course of 2D shock-water column interaction, using scheme 1. The maximum liquid density at time $t$ (---) and the minimum liquid density at time $t$ (---) evolve erratically, diverging significantly.](image1)

![Figure 4. Liquid density bounds during course of 2D shock-water column interaction, for schemes 2 through 5 (---, ---, ---, and ---, respectively). Dashed lines indicate maxima, and solid lines indicate minima.](image2)
prove on the curve from scheme 2, with each being more smooth and more horizontal than the previous. Though this complicated test case is ill-suited to determine concretely which treatment is the most accurate, it strongly suggests that scheme 5, our full treatment, is the most stable method for problems like this one.

\begin{figure}[h]
\centering
\includegraphics[width=0.5\textwidth]{figure5.png}
\caption{Minimum liquid pressure during course of 2D shock-water column interaction, for schemes 2 through 5 (---, ---, ---, and ---, respectively).}
\end{figure}

Summary and Conclusions
The treatment of the pressure, energy, and volume fraction in multiphase cells can be highly influential to the stability of a compressible, multiphase flow solver. In a framework that considers a sharp interface between phases and a pressure projection method, we have evaluated different approaches to calculations in multiphase cells and have recommended specific numerical improvements and innovations. This new approach has enabled us to simulate a liquid jet in supersonic crossflow, shown in Figure 6, with consistency and robustness that we could not otherwise attain.

References
Figure 6. Simulation of a liquid jet in supersonic crossflow. Liquid interface is depicted using the 0.5 isocontour of volume fraction. Center plane is colored by velocity magnitude, with 700 m/s as red and 0 m/s as blue. Snapshot taken at 1.73 ms.