Conservative Level Set/Ghost Fluid Method for Simulating Primary Atomization

O. Desjardins*, V. Moureau, E. Knudsen, M. Herrmann and H. Pitsch
Department of Mechanical Engineering
Stanford University, CA 94305

Abstract
This paper presents a novel approach for simulating incompressible two-phase flows by combining a conservative level set technique and the ghost-fluid method. Following the ideas of Olsson and Kreiss (2005), the liquid-gas interface is localized using a hyperbolic tangent level set that is transported and reinitialized using fully conservative numerical schemes. Mass conservation issues are greatly reduced in the process, therefore eliminating one of the major limitations of the level set technique. The ghost-fluid method provides a way of handling the interfacial forces and large density jumps associated with two-phase flows with good accuracy, while avoiding artificial spreading of the interface. This methodology is applied to simple test cases to assess its performance. We then use this technique to simulate the turbulent atomization of a liquid diesel jet.

*Corresponding Author
**Introduction**

A large number of fluid problems involve moving boundaries and interfaces. Applications range from air-water flows, reacting flows, and fluid-structure interactions, to crystal growth and image processing. For most of these applications, conservation of mass is critical to the stability and accuracy of the numerical results. Moreover, an accurate description of the interface location is often difficult to achieve, for resolution is typically limited. The turbulent atomization of a liquid jet may be one of the best examples to illustrate these challenges: The density ratio between the liquid and the gas is large, liquid mass conservation is paramount, and the competing physical processes lead to the formation of extremely thin liquid structures. These thin liquid structures are complex to model, for they carry a lot of momentum and have a strong influence on the turbulence. As a result, we want to limit ourselves to the direct numerical simulation (DNS) of all the scales involved in the atomization process. Such a simulation would provide some most needed information on how to accurately model the smallest scales, therefore bringing us one step closer to being able to perform large eddy simulation (LES) of such problems. Even with the premise that no subgrid-scale (SGS) model is required, we still face the challenge of finding numerical methods that can tackle the high density ratios, and provide stable and accurate solutions. Further, numerical schemes need to be capable of tracking the interface between liquid and gas at the limit of the numerical resolution, while maintaining good conservation properties.

In the past few years, several approaches have been used to overcome the problem of mass conservation in level set methods. Enright et al. [1] proposed a particle level set method (PLS), where Lagrangian markers are employed to correct the front location predicted by Eulerian transport. Sussman et al. [2] proposed to couple a level set method with a volume of fluid technique (CLSVOF) that has much better mass conservation properties. These methods have been quite successful, but typically suffer from an increased computational cost compared to a simple level set method, because of the number of particles per cell in the PLS method, and because an implicit formulation is impossible for CLSVOF, which limits the time-step size. Yet another common approach has been to refine the mesh locally in order to decrease the errors due to the level set transport. This refinement can be used for the level set equation only, such as in the case of the refined level set grid (RLSG) method [3], or it can be a standard arbitrary mesh refinement (AMR) approach, where the Navier-Stokes equations are also solved on the refined mesh [4]. This latter technique seems to be the only viable solution in case we are interested in many different scales. However, this method remains both extremely challenging to implement as well as extremely expensive. Moreover, in the context of primary atomization, the smallest scales of the interface might strongly affect the surrounding velocity field. Therefore, the difference between the minimum and the maximum level of refinement for such a case is very small, leading to little advantage of AMR compared to a uniformly refined mesh. The claim is often made that using a banded approach limits the cost of such methods by reducing the number of computational cells. However, this argument applies only for specific problems where the surface is not space filling. It can be contested for complex simulations where the average volume to surface ratio approaches the band width, such as the turbulent atomization of a liquid jet discussed below. In this case the amount of small scale structures and droplets surrounding the core of the jet is large. The interface is therefore likely to cover a large part of the computational domain, and thus would require the majority of the computational cells to be included in the band.

Recently, Olsson and Kreiss [5] proposed a conservative level set (CLS) method that can strongly reduce the problem of mass conservation with no additional cost compared to standard schemes. This methodology provides a low-cost alternative to the hybrid level set methods such as PLS and CLSVOF. However, the proposed CLS technique relies on a smeared out density jump as well as on spread out delta functions to represent the interfacial forces. This leads to a less accurate representation of the front instabilities and to a less robust formulation, prone to numerical instabilities such as spurious currents. In order to greatly reduce these issues, we will here combine CLS with the ghost fluid method (GFM) [6, 7], which treats the front as a sharp discontinuity, explicitly adding jump conditions to represent the surface tension force and the density jump across the interface. Combining the CLS approach with the GFM provides a low-cost and scalable level set method that has good mass conservation properties and retains a sharp description of the front. Such a method becomes therefore applicable in more complex geometries, and remains tractable for large scale problems, such as the DNS of an atomizing liquid jet.

In the next section, we will present the CLS methodology as well as some improvements that make the method robust in all situations. We will
then describe the GFM approach to handle the discontinuity. After presenting three test cases, namely the solid body rotation of Zalesak’s disk, a column in a deformation field, and a water droplet falling in a pool, we will finally address the simulation of a realistic liquid diesel jet.

**Conservative level set approach**

In order to limit mass conservation issues, the conservative level set approach proposed by Olsson and Kreiss [5] is employed here. However, in the case of highly turbulent fields and in the presence of frequent topology changes, the method is often unstable. A modified more stable version will be presented here.

**Standard level set formulation**

For incompressible multiphase flows, the continuity equation is often written in terms of the incompressibility condition

\[
\frac{D \rho}{Dt} = \frac{\partial \rho}{\partial t} + \mathbf{u} \cdot \nabla \rho = 0,
\]

where \( \rho \) is the density, \( \mathbf{u} \) the velocity, \( t \) the time, and \( D/Dt \) denotes the substantial derivative. In realistic applications, \( \rho \) can vary by several orders of magnitude across the interface, which does not violate (1), but which makes its numerical handling very difficult. Therefore, this equation is replaced by an equation for a smoother variable \( \phi \), from which the density can then be computed. Common practice is to choose \( \phi \) to be the signed distance from the interface, i.e.

\[
|\phi(x, t)| = |x - x_F|, \tag{2}
\]

where \( x_F \) corresponds to the closest point on the interface from \( x \), and \( \phi(x, t) > 0 \) on one side of the interface, and \( \phi(x, t) < 0 \) on the other side. With this definition of the level set function, the interface itself corresponds to the \( \phi(x, t) = 0 \) iso-surface. This choice leads to a very smooth field, which can be adequately transported and differentiated to compute the normal vector \( \mathbf{n} \) and the curvature \( \kappa \) of the interface defined as

\[
\mathbf{n} = \frac{\nabla \phi}{|\nabla \phi|} \tag{3}
\]

and

\[
\kappa = -\nabla \cdot \mathbf{n}. \tag{4}
\]

The main limitation of this approach is the absence of a built-in conservation property. Even on simple test cases, the mass loss can be dramatic [8], and therefore renders this method unsuitable for complex flows such as turbulent atomization.

**Original conservative level set formulation**

In order to improve the conservation property of level set methods, Olsson and Kreiss [5] proposed a novel approach based on transporting a hyperbolic tangent profile using a conservative formulation. This new level set function \( \psi \) can be defined as

\[
\psi(x, t) = \frac{1}{2} \left( \tanh \left( \frac{\phi(x, t)}{2\epsilon} \right) + 1 \right), \tag{5}
\]

and is transported by

\[
\frac{\partial \psi}{\partial t} + \nabla \cdot (\mathbf{u} \psi) = 0. \tag{6}
\]

Instead of defining the interface location by the iso-surface \( \phi = 0 \), it is now defined by the location of the \( \psi = 0.5 \) iso-surface. Provided that adequate numerical methods are used, and given the fact that the velocity field is solenoidal, the integral of \( \psi \) will be conserved exactly throughout the course of the simulation. This property obviously does not imply the conservation of the volume enclosed in the \( \psi = 0.5 \) iso-surface. However, it has a net tendency to improve the overall conservation property of the scheme. Of course, the numerical transport of the level set equation will lead to numerical diffusion of the hyperbolic tangent profile. Therefore, it is necessary to go through a re-initialization step that will re-sharpen the profile. This re-sharpening is also written in a fully conservative form, such that the combination of the transport step and the re-initialization step are conservative. The re-initialization step is expressed by

\[
\frac{\partial \psi}{\partial t} + \nabla \cdot (\psi (1 - \psi) \mathbf{n}) = \nabla \cdot (\kappa \nabla \psi). \tag{7}
\]

The approach that consists of solving (6) and (7) successively preserves the discrete conservation of the new level set function \( \psi \). If it is assumed that the interface is flat, it is straightforward to see that the conservation of \( \int \psi \) will automatically lead to the conservation of the volume enclosed in the \( \psi = 0.5 \) iso-surface. This, on the other hand, is not true anymore for a non-flat interface, since the thickness of the profile is non-zero. From this observation, it can be seen that smaller values of \( \epsilon \) will lead to better volume conservation. However, the profile still needs to be sufficiently resolved so that the transport step can be accurate and stable. In numerical experiments we have found that the value \( \epsilon = \frac{\Delta x}{2} \), where \( \Delta x \) is the mesh spacing, is satisfactory. This is in agreement with the value used by Olsson and Kreiss [5].

It should be emphasized again that the volume enclosed in the interface is not discretely conserved.
by this method. Olsson and Kreiss [5] directly used 
\( \rho(x, t) = \psi(\phi(x, t)) \), therefore allowing to claim \( \int \rho \) was conserved. However, this choice for the density 
severely smears out the density jump. This repre-
sents a major drawback, since the method might 
not accurately capture front instabilities. A great 
advantage of the conservative level set method is 
however, that the numerical errors caused by the 
level set transport will not be able to accumulate as 
much, which for the original level set method can 
lead quickly to a total loss of mass. This approach 
embeds some natural conservation property into the 
level set equation, which is sufficient to render com-
plex simulations possible.

**Improved conservative level set formulation**

The conservative level set technique presented 
in [5] presents significant improvements in term 
of mass conservation over standard level set ap-
proaches. However, it also suffers from some draw-
backs. The major issue that was observed is that 
the method is limited to simple problems with very 
smooth velocity fields. Indeed, because the hyper-
bolic tangent profile is contained within only two 
to three cells, the computation of the gradient of 
\( \psi \) is already a difficult procedure in general. Com-
bined with a complex topology that we can expect 
in turbulent break-up, this leads to severe inaccura-
cies when computing surface normals. Since these 
normals are of primary importance in the conserva-
tive re-initialization step, severe numerical instabil-
ities were observed. To improve the accuracy of the 
normal vector calculation, we modified the original 
procedure by first recomputing \( \phi \) from \( \psi \). Re-
constructing a standard signed distance function can 
be done efficiently using numerical methods such as 
fast marching [9]. With this additional step, we have 
at our disposal a much smoother scalar field, from 
which we can accurately extract surface normals and 
curvature. The final algorithm consists of the follow-
ing steps:

1. **Advance the level set function \( \psi \):** We use a 
third order conservative QUICK scheme to solve 
(6). The time integration is similar to the pop-
ular second order Crank-Nicolson, following the 
method described in [10].

2. **Re-initialize the distance function \( \phi \):** A fast 
marching approach is used to reconstruct a dis-
tance from the front. The initialization of the 
distance for the nodes closest to the front is ob-
tained by inverting (5).

3. **Compute normals \( \mathbf{n} \) from \( \phi \) field:** We use sec-
oond order central differencing to construct the 
normals from the \( \phi \) field.

4. **Apply conservative re-initialization:** Through-
out the convergence of (7), the normal vectors 
\( \mathbf{n} \) are kept constant. As observed by Olsson 
and Kreiss [5], this procedure converges very fast 
because of the presence of the diffusion term on 
the right-hand-side of (7).

**Ghost fluid methodology**

The level set formulation presented in the pre-
vious section provides an adequate way of identifying 
the location of the interface between liquid and 
gas. However, one still needs to account for interfa-
cial forces and large density ratios. The ghost fluid 
formulation [6, 7] provides an interesting solution 
by solving two continuous problems in the gas and 
in the liquid that are matched by satisfying jump 
conditions at the interface. By using explicit jump 
conditions, the spreading of the interface is avoided, 
leading to improved accuracy and stability.

**Navier-Stokes equation**

The Navier-Stokes equation is written as

\[
\frac{\partial \mathbf{u}_i}{\partial t} + \frac{\partial \mathbf{u}_i \mathbf{u}_j}{\partial x_j} = -\frac{1}{\rho} \frac{\partial p}{\partial x_i} + \frac{1}{\rho} \frac{\partial \tau_{ij}}{\partial x_j},
\]

where \( p \) is the pressure and \( \tau_{ij} \) is the viscous stress 
tensor defined by

\[
\tau_{ij} = \mu \left( \frac{\partial \mathbf{u}_i}{\partial x_j} + \frac{\partial \mathbf{u}_j}{\partial x_i} \right),
\]

and \( \mu \) is the dynamic viscosity.

The jump condition for the pressure contains the 
effect of the surface tension force as well as the effect 
of the discontinuity of the stress tensor normal to the 
interface, namely

\[
[p] = \sigma \kappa + 2[\mu] \mathbf{n}^\top \cdot \nabla \mathbf{u} \cdot \mathbf{n},
\]

where for any quantity \( \alpha \) we define its jump ac-
ross the interface by \([\alpha]\) = \( \alpha_l - \alpha_g \). The subscripts \( l \) 
and \( g \) indicate the liquid phase and the gas phase, 
respectively.

**Numerical formulation**

The formulation we use is based on a frac-
tional step method. The Navier-Stokes equations 
are solved without pressure gradient to obtain an 
intermediate velocity field, which is then corrected 
to enforce the divergence-free condition through the 
solution of a Poisson equation for the pressure. The 
variable arrangement is staggered in space and time. 
The level set function \( \psi \), the distance function \( \phi \), the
density $\rho$, as well as the pressure and the viscosity are cell-centered, and stored at time levels $t^{n-\frac{1}{2}}$ and $t^{n+\frac{1}{2}}$, while the velocities are face-centered and located at time levels $t^n$ and $t^{n+1}$. The algorithm consists of the following steps:

1. Advance the level set from time $t^{n-\frac{1}{2}}$ to $t^{n+\frac{1}{2}}$ using the velocity at time $t^n$ with the algorithm presented in the previous section.

2. Advance the momentum equation (8) from $t^n$ to $t^{n+1}$ without the pressure gradient: We use the fully conservative numerical scheme of Morinishi et al. [11] to discretize the spatial derivatives and the method of [10] for second order temporal integration.

3. Solve the Poisson equation using the GFM methodology [7]. A Krylov-based solver with multi-grid preconditioning is used.

4. Correct the velocity using the pressure gradient obtained using the GFM methodology [7].

**Examples**

This section presents several test cases to illustrate the capability of the proposed method. The first two test cases evaluate the tracking capabilities of the scheme. By varying the frequency of the conservative re-initialization step, the impact of the application of (7) on the quality of the level set transport can be assessed. We then apply the proposed approach on a three dimensional problem with topology change, namely a water droplet falling into a water pool. Finally, we address the turbulent breakup of a liquid diesel jet to demonstrate the ability of the method to simulate realistic problems.

**Zalesak’s disk**

A disk of radius 0.15 with a notch of width 0.05 and height 0.25 is placed in a $[-0.5, 0.5] \times [-0.5, 0.5]$ domain at $(0, 0.25)$. A solid body rotation is applied to the disk, and the shape of the notched circle is compared after one rotation. A thousand steps are used to perform the time integration. Figure 1 compares the shape obtained on a $100 \times 100$ mesh after one rotation with the conservative re-initialization step performed with different frequencies.

This test case lets us evaluate how much interfacial movement is induced by applying the conservative re-initialization. Indeed, it is clear that while re-sharpening the hyperbolic tangent profile, (7) will move the front. As a result, it is of primary importance that it is ensured that despite this shortcoming, the conservative re-initialization can be used without deteriorating too much the transport itself. From Fig. 1, one can observe that the more frequently (7) is applied, the worse the solution becomes. However, provided we perform the conservative re-initialization step every five time steps or less, the quality of the obtained shape appears satisfactory. One can even notice that the last case, where the conservative re-initialization step is performed only every ten time steps, gives better results than for no re-initialization at all. Figure 2 shows the evolution of the volume enclosed in the interface. Except for a small initial error that is caused by the fact that the sharp corners of the initial notched disk cannot be represented on the computational mesh, it appears that using the conservative re-initialization, regardless of the frequency, significantly improves the mass conservation property of the scheme. It will be shown later that a certain re-initialization frequency is required for more complex cases to conserve the mass. From this and the results shown in Fig. 1, it is concluded that applying (7) every five time steps is a reasonable choice, leading to good mass conservation properties as well as good accuracy in the transport.
Figure 2. Comparison of the time evolution of the volume enclosed in the notched disk; no re-initialization (○), re-initialization at each time step (+), every five time steps (×), and every ten time steps (□).

Column in a deformation field

The notched circle allowed us to gain some insights on the behavior of the proposed scheme. However, it is not a critical test case in terms of mass conservation. In order to further understand how the method actually improves mass conservation, we now consider a disk of radius 0.15 at (0.5, 0.75) in a unit box being stretched into a thin ligament by a velocity field given by the stream function

$$\Psi(x, t) = \frac{1}{\pi} \sin^2(\pi x) \cos^2(\pi y) \cos(\pi t/T). \quad (11)$$

This problem, introduced by Bell et al. [12], leads to a thin ligament eventually unresolvable by a given grid. The standard level set scheme will therefore loose some of the mass originally located in the circle. The velocity field defined by (11) leads to a maximum stretching at \( t = T/2 \) and then reverses, such that the original circle should be recovered at \( t = T \). We choose \( T = 8 \) and use two different grids \((128 \times 128 \text{ and } 256 \times 256)\). For both grids, the time step size is kept constant at \( \Delta t = 0.002 \), and we compare the results without re-initialization and with re-initialization performed every five time steps. Results are shown at \( t = T/2 \), where the stretching is maximum, and at \( t = T \), where the interface should be back to the initial circle. Figure 3 shows the interface shape at \( t = T/2 \) for the different runs. The main region where errors occur is the trailing edge of the filament. In the cases without conservative re-initialization, the tip of this trailing edge is lost by errors in the level set transport. The finer the mesh, the more the end of the filament is preserved. However, if re-initialization is performed, the tip of the filament is maintained in the form of drops. The coarser mesh leads to more larger drops, while the finer mesh gives only two very small drops and one detached ligament. This illustrates how the conservative re-initialization step can affect the interface location. By re-sharpening the level set function, it moves the interface until the structures can be resolved on the computational mesh, therefore introducing a cut-off length (namely, \( \epsilon \) in (5)), beyond which no structures are allowed to form. This process is of course not physically correct, but does not seem to be worse than allowing these structures to form only to remove them altogether by lack of resolution. This process could be understood as an SGS model that acts by introducing a surface tension-like force as soon as the interface becomes locally underresolved.

Figure 3. Interface shape of column in a deformation field at \( t = T/2 \); 128 × 128 mesh without re-initialization (upper left), 128 × 128 mesh with re-initialization every five time steps (upper right), 256 × 256 mesh without re-initialization (lower left), and 256 × 256 mesh with re-initialization every five time steps (lower right).

Figure 4 presents the solutions that are obtained with the different methods at \( t = T \). Clearly, omitting the re-initialization leads to serious volume loss, especially on the coarser mesh. However, while the volume seems better preserved with reinitialization,
one can clearly see the consequence of having generated the drops at the tip of the trailing edge. The level set transport is strongly affected, leading to large errors in the recovered shape for the $128 \times 128$ mesh. The finer mesh gives acceptable accuracy by giving both good conservation and good shape-preservation. Finally, Fig. 5 quantifies the amount of volume that is lost throughout the simulations. While the two cases without re-initialization both display large errors in the final volume, the cases with re-initialization show a largely improved behavior, with errors in volume conservation at least one order of magnitude lower.

![Figure 4](image4.png)

**Figure 4.** Interface shape of column in a deformation field at $t = T$; exact solution (thin line) and $128 \times 128$ mesh without re-initialization (thick line, upper left), $128 \times 128$ mesh with re-initialization every five time steps (upper right), $256 \times 256$ mesh without re-initialization (lower left), and $256 \times 256$ mesh with re-initialization every five time steps (lower right).

**Falling water droplet**

In order to evaluate the ability of the methodology to handle realistic problems with topology changes and large density ratios, a water droplet of diameter $D = 1$ cm falling in a water pool of height $H = 1$ cm is computed. The density of the water is $\rho_l = 1000$ kg·m$^{-3}$, and the density of the air is $\rho_g = 1.226$ kg·m$^{-3}$, resulting in a density ratio of $\rho_l/\rho_g = 815.66$. The viscosity of the liquid is taken to be $\mu_l = 1.137 \times 10^{-3}$ kg·m$^{-1}$·s$^{-1}$, while the viscosity of the air is $\mu_g = 1.137 \times 10^{-3}$ kg·m$^{-1}$·s$^{-1}$. The surface tension is set to $\sigma = 0.0728$ N·m$^{-1}$, and the acceleration due to gravity is $g = 9.81$ m·s$^{-2}$. The droplet is initially at rest at one centimeter above the pool. The computational domain is a four centimeters wide cubic region with periodic boundary conditions on the sides, a free-slip condition at the top boundary, and a wall at the bottom. A $256^3$ mesh is used to perform the computation. Figure 6 shows the evolution of the simulation. We observe that the initial symmetry is lost when the droplet impacts the pool. A small irregular liquid crest is formed on the side of the droplet. As this crest retracts due to surface tension, a liquid jet is formed at the center of the domain. This water ligament eventually retracts as well because of the gravitational force, and ejects a few small size droplets at its tip. Finally, the simulation enters a simple oscillatory mode while the last droplets start falling. The proposed methodology proves to be very robust on this problem. Moreover, the cost of the level set transport, re-initialization (conservative and distance), and GFM represent less than 20% of the overall simulation. As expected in an incompressible solver, the pressure solver remains the most expensive element.

**Turbulent atomization of a liquid diesel jet**

The main goal of this work is to investigate the turbulent atomization of a liquid jet. Using the proposed numerical methodology to handle the liquid-
gas interface, we simulate the atomization process of a liquid diesel jet. The properties of the two phases are summarized in Table 1.

<table>
<thead>
<tr>
<th>Phase</th>
<th>Density (kg·m⁻³)</th>
<th>Viscosity (kg·m⁻¹·s⁻¹)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Liquid</td>
<td>850</td>
<td>1.70 × 10⁻³</td>
</tr>
<tr>
<td>Gas</td>
<td>25</td>
<td>1.78 × 10⁻⁵</td>
</tr>
</tbody>
</table>

Table 1. Properties of the two phases used to compute the turbulent atomization of a liquid diesel jet.

A 2 mm × 0.5 mm × 0.5 mm computational domain is discretized using a 512 × 128 × 128 mesh. A circular liquid jet of diameter $D = 0.1$ mm is injected into the domain. The injection velocity is set to $U_0 = 100$ m·s⁻¹, while the gas velocity at the inlet plane is zero. At the exit of the domain, we employ a convective outflow condition for both the velocity and the level set function. At the lateral boundaries, we enforce a periodicity condition. The simulation is initialized from a half-sphere of liquid at rest at the injection plane.

Two snapshots of the liquid jet at an intermediate and at the final time are presented in Fig. 7. We can first notice that despite the fact that no turbulent inflow conditions are used, the liquid jet is fully turbulent. In the intermediate frame, most of the droplets have been generated by shedding of the mushroom shape at the tip of the jet. However, in the later frame, generation and break-up of ligaments occur in many different places throughout the whole length of the jet. A zoom on the middle part of the jet is presented in Fig. 8. The surface topology appears extremely complex, composed of stretched-out ligaments and droplets. However, a clear liquid core can be seen behind these structures. Many different droplet sizes and shapes are obtained, ranging from the smallest possible droplets on the computational mesh that we used, to large liquid blobs more than ten computational cells wide.

In diesel engines, the turbulence generated by the injection of the liquid fuel is of great importance for the combustion process and the formation of pollutants, because it strongly affects the rate of mixing in the cylinder. In order to gain some understanding of the mechanisms by which turbulence is induced in the gas surrounding the liquid jet, we show the Q criterion [13] superimposed on the interface position in Fig. 9. The Q criterion, which is the second invariant of the velocity gradient tensor, was proven to provide a good way to identify coherent turbulent structures [14]. The liquid jet tip, while penetrating in the gas at rest, generates annular vortices that are torn up into horseshoe-like vortices as they move away from the liquid core. These vortical structures interact among each other and with the liquid ligaments and droplets surrounding the jet, leading to fully developed turbulence. The complex interaction that can be observed between the droplets and the vortices indicates how important it is for the level set method not to loose these small scale structures because of mass conservation issues.

The numerical uncertainties in this simulation are numerous. Future work towards achieving a reliable DNS of the turbulent atomization of a liquid jet should include a mesh refinement study in order to assess whether the physical mechanisms are accurately represented in this simulation.

**Conclusion**

A conservative level set/ghost fluid method has been presented that combines the excellent conservation properties of the conservative level set proposed by Olsson and Kreiss [5] with the sharp interfacial representation of the GFM. This method proves to be efficient, robust, and relatively accurate, with the additional benefit of good mass conservation properties. Using this approach, the turbulent atomization of a liquid diesel jet was...
simulated. While more work is required to prove that the numerical methods allow for an accurate description of all physical processes, this simulation can shed some light on the complex phenomena behind turbulent atomization, therefore providing most needed help toward developing SGS models to be used in realistic simulations.

References


Figure 7. Two successive snapshots of the turbulent atomization of a liquid diesel jet.
Figure 8. Zoom into the chaotic structure of the interface of the atomizing liquid diesel jet.

Figure 9. Q criterion (red iso-surface) superposed on the liquid jet (blue) being atomized.