Mixing in Converging Flows

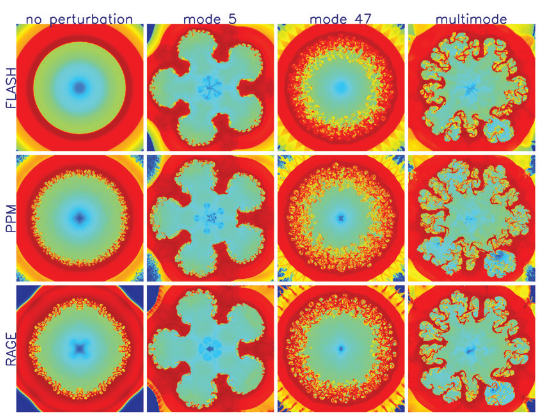
Candace C. Joggerst, T-2; Anthony Nelson, University of North Carolina; Paul Woodward, University of Minnesota; Catherine Lovekin, T-2; Thomas Masser, CCS-2; Guy Dimonte, XCP-5; Christopher L. Fryer CCS-2; Praveen Ramaprabhu, University of North Carolina; Marianne M. Francois, Gabriel M. Rockefeller, CCS-2

The growth of turbulence in converging flows can affect scenarios as small as the implosion of inertial confinement fusion pellets and as large as the collapse and explosion of massive stars. Such scenarios are often challenging to model, both from a purely hydrodynamic perspective and because they often require accurate simulation of many different physical processes. We present initial results of an investigation into turbulence and mixing in converging flows, using four different codes to simulate cylindrical and spherical implosions on 2D meshes. The results of these simulations lay the groundwork for future 3D studies.

In a problem involving the implosion of a sphere or cylinder, Rayleigh-Taylor and Richtmyer-Meshkov instabilities may both be present. The Rayleigh-Taylor instability develops when the density and pressure gradients in a fluid are opposed, as when a denser fluid rests atop a lighter fluid in a gravitational field, or when a lighter fluid accelerates a denser fluid. The Richtmyer-Meshkov instability develops when a shock passes through an interface between a heavy and a light fluid.

Most simulations and physical experiments that study the details of the growth of these instabilities have been performed in planar geometry (for example, see [1]). However, Rayleigh-Taylor and Richtmyer-Meshkov instabilities also play important roles in non-planar problems. Instability growth on such disparate scales as inertial confinement fusion capsules and the interiors of pre-supernova stars may dramatically influence the evolution of these implosions [2].

Fig. 1. Simulated 2D cylindrical implosions of dense shells of fluid surrounding relatively light cores, near the time of maximum compression.



Ideally, simulations of physical phenomena should minimize the presence of artifacts of the numerical method. Simulations of instabilities in radially converging flows present extra challenges beyond those present in simulations of instabilities in planar interfaces; in particular, the ideal spatial coordinate system and mesh are no longer obvious. While spherical coordinates may seem appealing for following the flow of fluid in spherical implosions, the simplicity of mesh-aligned flow is lost once turbulence develops. Spherical coordinates also introduce coordinate singularities that impose a preferred orientation on the simulation, and features may develop differently along the axis than at other points around the sphere. In multiphysics problems, such features can expand and contaminate regions far from the axis. Cartesian coordinates avoid any coordinate singularities and are suitable for simulating a wide variety of physical problems, but using them to simulate spherical implosions raises questions about the spatial resolution required not just for simulating curved interfaces on rectilinear meshes, but also for following the growth of perturbations imposed on those interfaces.

Youngs and Williams [3] used a Lagrange-remap code on a 3D spherical polar mesh to simulate turbulent mixing in a sector of a spherical implosion with random perturbations that were initially applied to the interface between light and dense fluids. The authors found that the width of the mixing zone shrank slightly as they increased the resolution of the mesh but that, at the finest mesh resolution they used, the mixing zone width seemed to have converged.

Do simulations in other coordinate systems, on other meshes, achieve similar levels of convergence? Specifically, what resolution would be required for a simulation of a spherical implosion on a Cartesian mesh to yield a converged measurement of the mixing zone width? Do other diagnostics of mixing and turbulence demonstrate the same degree of convergence as mesh resolution increases?

For this project, we compare results of simulations of radially converging flows from four codes: RAGE [4], CASTRO [5], FLASH [6], and multi-fluid PPM [7]. All four codes model compressible hydrodynamics on Eulerian meshes, but they differ in the details of their discretization schemes, their shock-capturing methods, limiters and steepeners, their treatment of materials in mixed cells, and their support for different coordinate systems. We assess the impact of the choice of coordinate system on the ability of these codes to follow the growth of turbulence from perturbed interfaces, and we have begun to quantify

For more information contact Candace C. Joggerst at church@lanl.gov

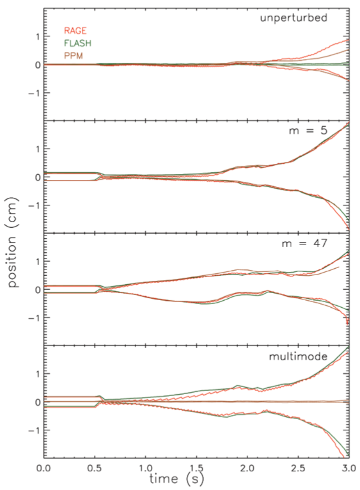


Fig. 2. Azimuthally averaged heights of bubbles of light fluid, and depths of spikes of dense fluid, as a function of time in the cylindrical implosions.

the spatial resolution required to follow perturbations with different wavelengths and initial amplitudes.

Figure 1 shows snapshots of simulated 2D cylindrical implosions of dense shells of fluid surrounding relatively light cores, near the time of maximum compression. Color represents the mass fraction of the dense fluid–the reddest regions contain unmixed dense fluid, while the bluest regions contain unmixed light fluid, and yellows and greens indicate regions where substantial mixing has occurred. The interface between light and dense fluids was either left unperturbed (left-most column) or initialized with one of three different perturbations—a long-wavelength perturbation (mode 5), a short-wavelength perturbation (mode 47), or a multi-mode spectrum.

Results from the three perturbed-interface simulations compare well across all codes. This degree of agreement is particularly interesting considering the different mesh geometries and refinement strategies employed by the codes: the RAGE calculations used a Cartesian mesh with adaptive mesh refinement, the FLASH calculations used a polar mesh with adaptive mesh refinement, and the PPM calculations used a uniform Cartesian mesh that moved inward with the flow. Differences appear at small scales–for example, in the specific pattern of secondary instabilities at the edges of the inward spikes of dense fluid in the mode-5 calculations. Using a polar mesh, FLASH is able to preserve an unperturbed cylindrical interface, while PPM and RAGE (using Cartesian meshes) introduce perturbations at the mesh scale as the interface moves inward. However, results on the Cartesian meshes compare well with results on FLASH's polar mesh once initial perturbations are introduced.

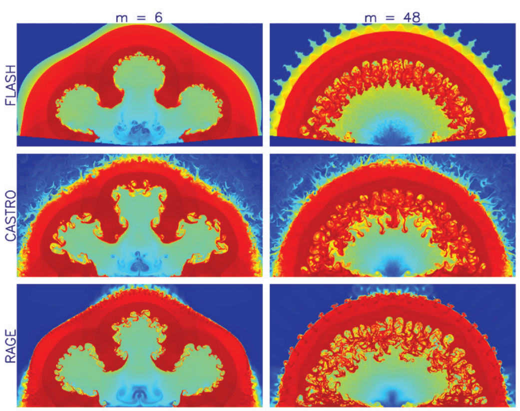


Fig. 3. Snapshots of simulated spherical implosions.

Figure 2 shows the azimuthally-averaged heights of bubbles of light fluid, and depths of spikes of dense fluid, as a function of time in the cylindrical implosions. Again, FLASH's polar mesh introduces no perturbations into an initially unperturbed implosion. Bubble and spike positions agree well across all three codes in the perturbed calculations, although the degree of agreement sometimes varies with time.

Figure 3 shows snapshots of simulated spherical implosions, using the same color scheme as Fig. 1. Because the simulations were performed in 2D RZ coordinates, only half of the sphere was represented, and the interface between light and dense fluids was initialized with even-mode-number perturbations (modes 6 and 48). Bubble and spike amplitudes are again similar across the three codes, but differences in the detailed development of turbulent features are apparent–for example, small-scale features are generally narrowest and sharpest in RAGE.

Ultimately, our goal is to simulate mixing and turbulence in full 3D scenarios. Results from the simulations presented here help us to quantify the spatial resolution required to follow the growth of perturbations of a given initial amplitude and wavelength. The degree of agreement among results on different mesh geometries from codes using different hydrodynamics algorithms establishes requirements for simulating these scenarios in three spatial dimensions. The differences that do appear, in small-scale structure and in specific diagnostics of mixing, could provide specific targets for detailed numerical investigations or future physical experiments.

[1] Dimonte, G. et al., *Phy Fluid* **16,** 1668 (2004).
[2] Arnett, W.D. and C. Meakin, *Astrophys J* **733,** 78 (2011).
[3] Youngs, D.L. and R.J.R. Williams, *Int J Numer Meth Fluid* **56,** 1597 (2008).
[4] Gittings, M. et al., *Comput Sci Discov* **1,** 015005 (2008).
[5] Almgren, A.S. et al., *Astrophys J* **715,** 1221 (2010).
[6] Fryxell, B. et al., *Astrophys J Suppl* **131,** 273 (2000).
[7] Porter, D.H. and P.R. Woodward, "Simulating Compressible Turbulent Flow with PPM," *Implicit Large Eddy Simulation: Computing Turbulent Fluid Dynamics,* Cambridge University Press (2010).

Funding Acknowledgments
DOE NNSA, Advanced Simulation and Computing Program; LANL Laboratory Directed Research and Development