Investigation of Hydrodynamic Instabilities in Shock-Accelerated Flows for ICF

D. Ranjan, B. Motl, J. Niederhaus, J. Oakley, M. Anderson and R. Bonazza

Department of Engineering Physics, University of Wisconsin–Madison, Madison, WI 53706
manderson@engr.wisc.edu

In inertial confinement fusion (ICF) implosions, the development of a mixing layer at the fuel/ablator interface is driven by shock-induced hydrodynamic instabilities. These instabilities cause perturbations on the interface to grow in amplitude, deform, and eventually develop into turbulent mix. Because such behavior can result in substantial mixing between the compressed hot fuel and outer cold fuel and ablator material, the efficiency and yield of the reaction can be degraded. Thus, it becomes important to understand the physics of these instabilities, and, in particular, the amplitude growth and mixing rates, in order to design the capsules to ensure maximum possible burn efficiency.

Current experiments at the Wisconsin Shock Tube Laboratory (University of Wisconsin-Madison) are investigating these instabilities using various types of perturbed material interfaces in a strongly-shocked gas medium. These studies involve the measurement of instability growth rates, characterization of the unstable flows, and correlation of these characteristics with experimental parameters such as shock strength, material density ratio, and interface geometry and dimensions. These are made possible mainly by planar laser induced fluorescence (PLIF) in one gas species. Experimental studies are coupled with computational modeling that is used to optimize experiment design and to predict and analyze experimental results along with feed back to the computation models.

Recent experiments with a spherical soap bubble have been performed in the vertical, square, large internal cross section (25 x 25 cm) shock tube. The initial interface is three-dimensional, with axial symmetry being a reasonable assumption for a free falling bubble. The strength of the accelerating shock is in the range $2<M<4$. The interface is imaged once immediately before shock arrival and twice after interaction with the shock wave, so that the initial conditions are known and the growth rates can be calculated for each experiment.

Concurrently, the *Raptor* code is being used to numerically simulate the shock tube runs for the purpose of optimizing the design of the experiments and for comparing the experimental and numerical results. The code solves the compressible Navier-Stokes equations using a piecewise linear method (PLM) combined with adaptive mesh refinement (AMR).

This paper discusses the progress of the experiments and the computational effort. Examples of the post shock images are shown below.

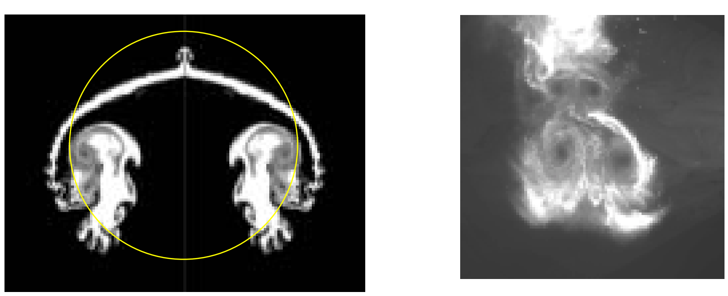

Figure 1. Computational and experimental image of a shocked bubble 0.99ms after impact with a 2.14 Mach shockwave.