MODELING OF MULTIPHASE FLUID-STRUCTURE INTERACTION PHENOMENA USING A COUPLED CFD/CSD METHODOLOGY

J. D. Baum[1], O. A. Soto[1], F. Togashi[1] & R. Lohner[2]

[1]Center for Applied Computational Sciences,
Science Applications International Corporation, 1710 SAIC Dr., MS 2-6-9
McLean, VA 22102, USA
[2]CFD Center, Dept. of Computational and Data Science, MS 6A2,
College of Science, George Mason University,
Fairfax, VA 22030, USA

The development of a coupled CFD/CSD methodology has been driven by the requirement to simulate structural response to blast loading, the flow field modifications due to the structural deformation and the downstream impact by the modified flow field and the deformed/fragmented structure. Consider weapon detonation outside a structure, where the interest is in modeling the exterior wall failure, and the interior loads exerted by for both blast and broken concrete/glass/bricks. Recently, we expanded the methodology to model blast interaction with multiphase materials, such as structures filled with water or foam, to enable modeling of the structural failure and the water/steam/droplet jetting.

Flow solvers for gases and liquids are combined via immersed body techniques to simulate liquid/gas multiphase problems. Both solvers are run concurrently. In the gas (i.e. compressible flow) region the velocities of the liquid are imposed wherever liquid is present. For the liquid (near-incompressible (+VOF)) region, the pressures of the gas region are imposed wherever gas is present. This multiphase flow solver is then linked to a structural mechanics solver to calculate the effects of blasts on structures with fluid dampers.

The final paper focuses on a study modeling the response of water-filled tubes to high-speed plate impact, with the objective of optimizing impact energy dispersion (i.e., stop the plate).