Direct numerical simulation of compressible multiphase flow using a discontinuous Galerkin based multiscale approach.

F. Jaegle*, S. Fechter, M. Boger, C. D. Munz
Institute of Aerodynamics and Gas Dynamics, University of Stuttgart, Germany
{jaegle,fechter,boger,munz}@iag.uni-stuttgart.de

Abstract

Many scientific or industrial applications for the direct simulation of multiphase flow warrant the use of a compressible formulation, as high temperatures and pressures or even shock waves may be encountered. Examples are rocket engines or automotive injection systems. However, the simulation of compressible multiphase flow is in certain respects more complex than the incompressible treatment that is often found in numerical solvers today.
Typically, two-phase flow solvers rely on two crucial elements: the first is a method that allows to define the geometry and the temporal evolution of the interface between the two phases. In the present study, a level set formulation is used. The second is a numerical strategy to treat the discontinuous nature of the interface as well as the related physics such as impinging waves, phase change or surface tension.
Mainly for incompressible flows, diffuse interface methods that allow numerical smearing of the interface are common. For compressible flow, this approach can be problematic as simple averaging of the equations of state (EOS) may not be consistent with the intermediate states. The alternative that we adopt in the present study is a sharp interface approach, which keeps the different fluids separate from a numerical perspective. Our multiscale method solves the Navier Stokes equations for the macroscopic scales of the flow. At the interface, jump conditions are provided by a separate solver for the microscale. Many microscale solvers are conceivable, in the present study, we rely on Riemann-type solvers. The approach is suitable for general EOS and has been tested for different formulations. In the present study, results for the combination of an ideal gas and the Tait EOS are presented.
The numerical method consists of the following components: for the macroscale solver, a discontinuous Galerkin (DG) spectral element method is used. This scheme does not enforce continuity between the elements, therefore, it relies on numerical fluxes similar to a finite volume scheme. In the bulk phases, numerical fluxes are obtained from a standard Riemann solver. The level set equation is solved using the same DG scheme. Near the phase interface, the jumps in certain flow variables are assumed to coincide with the nearest element boundary, where the Riemann solver is replaced by the microscale solver. Instead of evaluating a single numerical flux, the microscale solver yields discontinuous fluxes, which are applied on the liquid and gaseous side respectively. This procedure allows to conserve a sharp discontinuity at the interface. It is in a certain sense comparable to the well-known ghost fluid method. The model for surface tension is included in the microscale solver. It therefore takes its effect on the macroscales via the numerical fluxes, which eliminates the need for volume source terms to apply the surface force, as for example in the continuum surface force approach.
The DG scheme allows high orders of accuracy with subcell resolution, which is particularly valuable for the calculation of interface curvature that can be obtained from the derivation of the ansatz polynomials for the level set variable. However, high polynomial degrees are usually combined with large grid cells - the error introduced through the assumption of the interface coinciding with element faces therefore becomes large in configurations, where the DG scheme is most efficient. To avoid this contradiction, an adaptive grid refinement strategy is used in the vicinity of the interface. This framework allows to replace the high-order spectral elements by zones with very fine grids and first order accuracy. This is limited to the flow equations near the interface, where it increases the accuracy of the interface treatment and in addition guarantees oscillation-free results in the presence of shocks. At the same time, the level set remains unrefined, retaining the ease of curvature calculation from polynomial data.
The method has been validated for one-dimensional tests such as multiphase shock-tube problems in the past. We present several three-dimensional tests. For a quantitative validation, the interaction between a droplet or a bubble and a concentric shock wave are simulated in 3D and compared to well-resolved reference solutions obtained with a 1D solver assuming spherical symmetry. In addition, we present a 3D interaction between a planar shock and a spherical droplet, which shows the typical shock reflections and deformations in- and outside the droplet.

---

*Corresponding author: felix.jaegle@iag.uni-stuttgart.de