11th World Congress on Computational Mechanics (WCCM XI)
5th European Conference on Computational Mechanics (ECCM V)
6th European Conference on Computational Fluid Dynamics (ECFD VI)
July 20–25, 2014, Barcelona, Spain

HIGH-RESOLUTION NUMERICAL METHOD FOR COMPRESSIBLE LARGE-EDDY SIMULATION OF CAVITATING LIQUID FLOWS

Christian P. Egerer*, **Steffen J. Schmidt**,
Stefan Hickel and Nikolaus A. Adams

Institute of Aerodynamics and Fluid Mechanics, Technische Universität München
Boltzmannstr. 15, 85748 Garching bei München, Germany
(*) *christian.egerer@aer.mw.tum.de*, *http://www.aer.mw.tum.de*

Key words: *Cavitation, Large-Eddy Simulation, Numerical Methods*

Cavitation plays an important role in many engineering and medical applications, e.g., liquid fuel injection systems, hydraulic machines, or extracorporeal shock wave lithotripsy. Wave as well as turbulence dynamics need to be incorporated in numerical solution strategies, since shock waves are emitted and propagate through the liquid-vapor mixture during collapse of vapor cavities, and cavitation and turbulence dynamics can occur on same time and length scales. Thus, numerical methods capable of capturing discontinuities in the flow field, on the one hand, and capable of resolving broad-band turbulence, on the other hand, are required.

When viscous effects are neglected, single-fluid cavitation models based on the homogeneous mixture assumption have been proven to give reliable predictions of hydrodynamic and wave dynamics effects in cavitating liquid flows. A finite volume method solving the Euler equations for a homogeneous mixture of liquid and vapor has been developed in our in-house code CATUM [1]. We present a high-resolution extension to CATUM suitable for compressible large-eddy simulation of cavitating liquid flows. A sensor functional based on Ducros's shock sensor or the variation of the vapor volume fraction is used to blend between a central non-dissipative reconstruction of primitive variables at cell faces and an upwind-biased reconstruction capable of capturing shock waves and pseudo-phase-boundaries with large density gradients.

Standard test cases demonstrate the suitability of the present numerical method for problems involving shock and expansion waves, evaporation and turbulence. Fig. 1 compares the present scheme to a reference solution for Test 1 of Toro's textbook [2] on a homogeneous grid with $N = 200$ cells. Phase change due to cavitation in a 1-D expansion tube is considered in Fig. 2 also with $N = 200$ cells. Statistical quantities and spectra for decaying compressible isotropic turbulence with initial Reynolds number $Re_\lambda = 100$ and turbulent Mach number $Ma_t = 0.3$ on a grid with 32^3 cells is presented in Fig. 3.

Further validation will be performed by applying the present method to a 3-D single bubble collapse and a 3-D bubble cloud collapse and comparison with inviscid solutions [3].

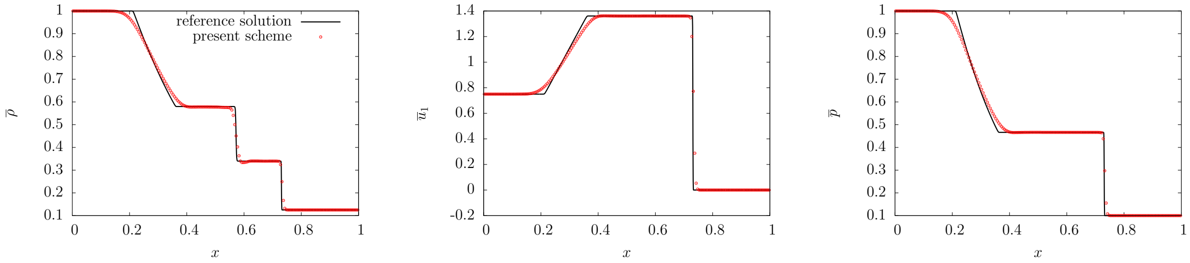

Figure 1: Shock tube problem after $t = 0.2$ with $N = 200$ cells: density (left), velocity (center), and pressure (right).

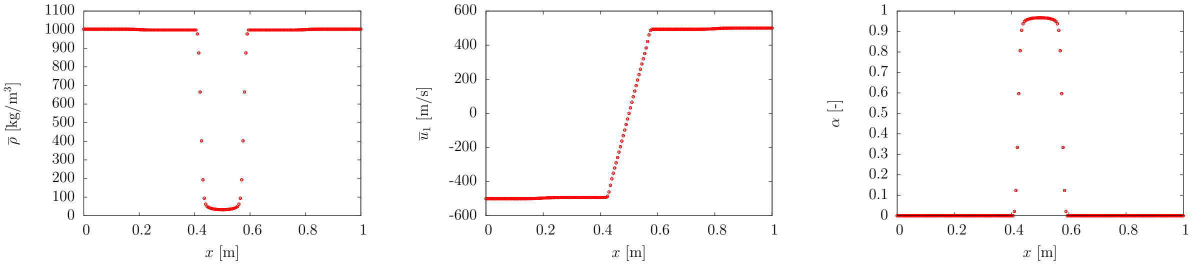

Figure 2: Double expansion wave with phase-change after $t = 0.15 \times 10^{-3}$ s with $N = 200$ cells: density (left), velocity (center), and vapour volume fraction (right).

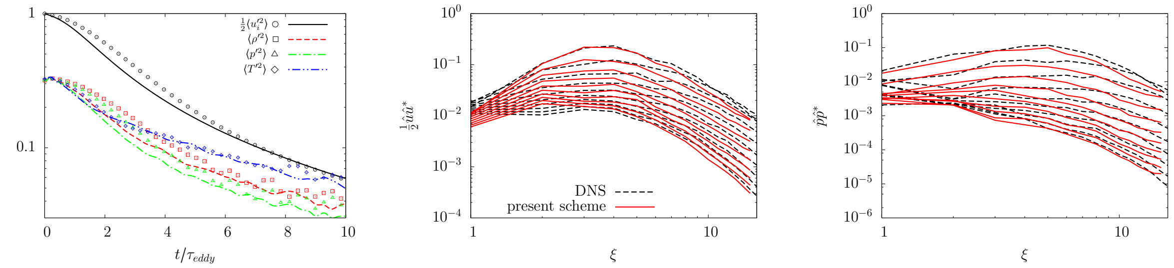

Figure 3: Comparison of DNS (symbols) and LES (lines) for decaying compressible isotropic turbulence with $Re_\lambda = 100$ and $Ma_t = 0.3$: time evolution of normalised fluctuating quantities (left), spectra of turbulence kinetic energy (center), and spectra of pressure (right).

REFERENCES

[1] G. H. Schnerr, I. H. Sezal, and S. J. Schmidt. Numerical investigation of three-dimensional cloud cavitation with special emphasis on collapse induced shock dynamics. *Phys Fluids*, Vol. **20**, 040703, 2008.

[2] E. F. Toro. Riemann Solvers and Numerical Methods for Fluid Dynamics – A Practical Introduction. Third Edition, Springer, 2009.

[3] N. A. Adams, S. J. Schmidt. Shocks in Cavitating Flows in *Bubble Dynamics and Shock Waves*, p. 235-256, Springer, 2013.