An accurate inviscid compressible solver for aerodynamic applications

G. Romanelli, E. Serioli and P. Mantegazza

Dipartimento di Ingegneria Aerospaziale, Politecnico di Milano, Italy

Abstract

A new OpenFOAM cell-centered Finite Volume solver, called AeroFoam, for the accurate numerical simulation of multi-dimensional time-dependent inviscid compressible fluid flows on polyhedral unstructured meshes is presented.

A fully coupled explicit formulation is preferred to the semi-implicit segregated formulation implemented among the existing OpenFOAM time-dependent inviscid compressible solvers, such as sonicFoam, rhoSonicFoam and rhopSonicFoam. The conservative flux function vector for the Euler equations is approximated with a number of the flux splitting numerical schemes presented in Literature, from the *Convective Upwind Split Pressure* (CUSP) by Jameson to the more accurate and computationally expensive *Approximate Riemann Solver* (ARS) by Roe. To achieve second order accuracy in space and yield high resolution in presence of discontinuities, limited flux splitting numerical schemes such as *Lax-Wendroff*'s (LW) or *Jameson-Schmidt-Turkel*'s (JST) are also implemented, after an appropriate extended cells connectivity is built in the pre-processing stage, for instance with meshSearch utility functions. Both standard and TVD low storage explicit Runge-Kutta methods of order $p = 2, 3, 4$ are choosen as a compromise between the computational efficiency, the memory requirements, the *Courant-Friedrichs-Lewy* (CFL) stability condition and the order of accuracy in time of the numerical solution.

To evaluate the new solver performances, several 2D and 3D, subsonic, transonic and supersonic benchmark test problems are tackled. Despite being explicit with no convergence acceleration techniques, e.g. multigrid, AeroFoam shows an improvement in accuracy and computational efficiency with respect to the aformentioned OpenFOAM solvers, and its performances are close to those of the explicit time-dependent inviscid compressible coupled solvers of commercial software such as FLUENT (without multigrid).

Some challenging problems are also investigated, such as the study of simple fluid-structure interaction (FSI) phenomena (airfoil pitching and plunging). In this case the body movement is simulated by means of transpiration boundary conditions, while the mesh is actually deformed only in the post-processing stage with dynamicMesh utility functions.

AeroFoam is currently being used to solve more complex 3D problems, from the numerical simulation of the flow field around a complete aircraft (YF-17 fighter) to the computation of flutter boundary for an aeroelastic benchmark test case (AGARD 445.6 wing).

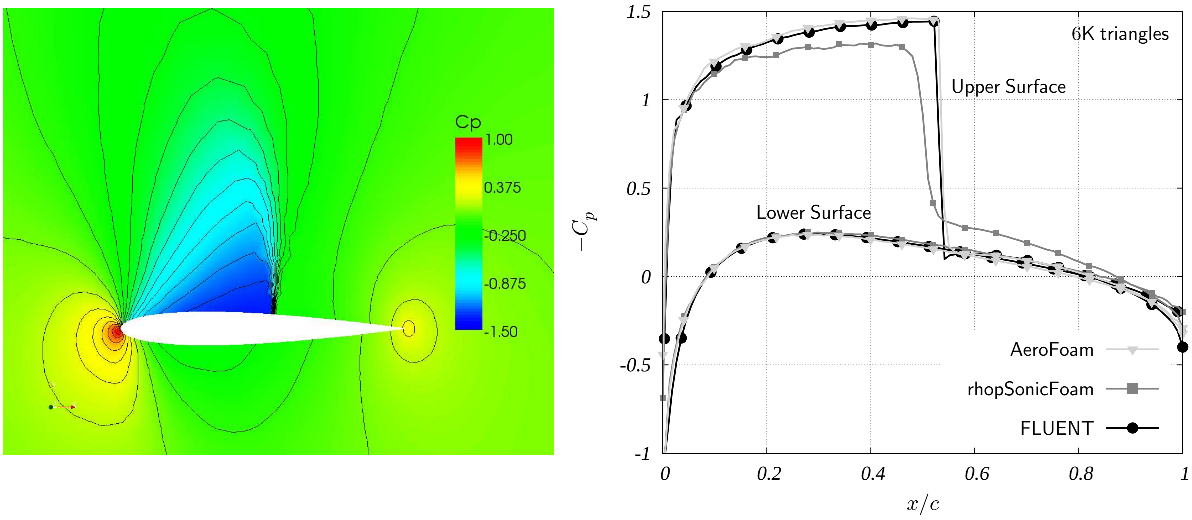

Figure 1: Transonic flow around the NACA 0012 airfoil at $M_\infty = 0.75$ and $\alpha = 4.00°$ computed with AeroFoam. On the left the isolines of the pressure coefficient C_p are shown. On the right the distribution of the pressure coefficient C_p along the airfoil chord c computed with AeroFoam (▼) is compared with rhopSonicFoam (■) and FLUENT (●) results.

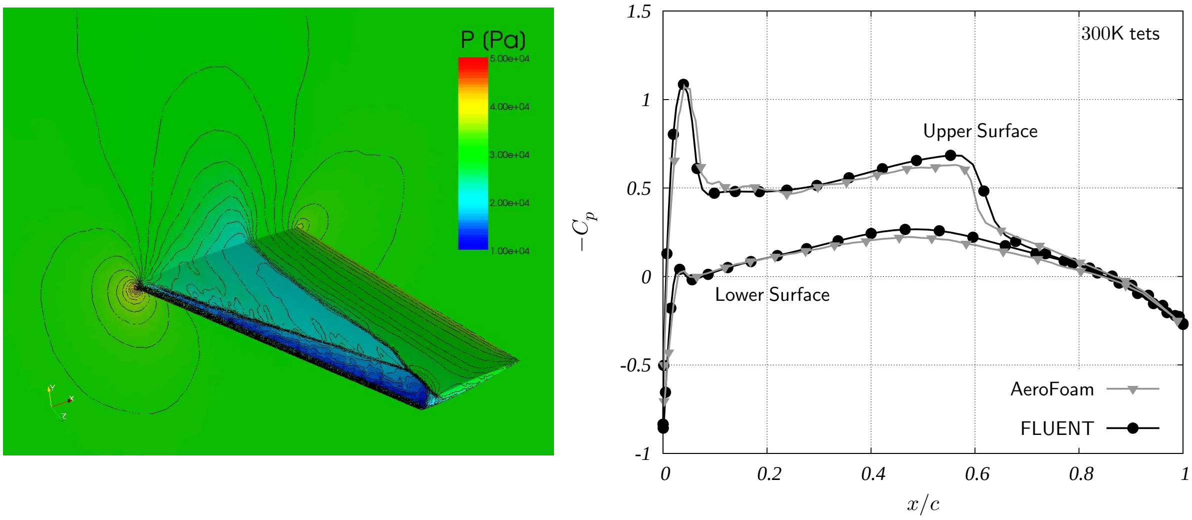

Figure 2: Steady-state transonic flow around the ONERA M6 wing at $M_\infty = 0.84$ and $\alpha = 3.06°$ computed with AeroFoam. On the left the isolines of the thermodynamic pressure P on the wing and the symmetry plane are shown. The λ-shaped shock wave is accurately resolved. On the right the distribution of the pressure coefficient C_p at the wing span station $y/b = 0.2$ computed with AeroFoam (▼) is compared with FLUENT (●) results.