Computation of compressible flows on unstructured staggered grids

I. Wenneker, A. Segal and P. Wesseling

J.M. Burgers Center and Faculty of Information Technology and Systems
Delft University of Technology, Mekelweg 4, 2628 CD Delft, Holland
E-mail: I.Wenneker@math.tudelft.nl
Supported by The Netherlands Organization for Scientific Research (NWO)

Abstract

A numerical scheme to compute 2D inviscid compressible flows on unstructured grids using a staggered positioning of the variables is presented. A finite volume scheme is used to discretize the conservation laws.

We restrict ourselves to unstructured grids that consist of triangles. At the triangle centroids the scalar variables are located. At the midpoints of the edges the normal momentum component is positioned. This placement of the variables, see Figure 1, is similar to the classic staggered scheme on structured grids with quadrilateral cells. Use of this staggered grid facilitates future extension to weakly compressible and incompressible flows.

As primary variables we have chosen the density ρ, the momentum vector $\mathbf{m} = \rho\mathbf{u}$ and the energy variable ρH, whereas the third thermodynamic variable (in our case the pressure p) follows from the equation of state. The equations are decoupled in such a way that each primary variable is solved for one after another. Hence, in this way each of the conservation laws is considered (more or less) as an independent convection equation. Such a segregation of the equations is common in the field of the shallow water equations, but it is seldom encountered in the field of aerodynamics. An implicit Euler scheme is employed to do the time stepping. This is done in such a way that both stationary as nonstationary problems can be solved in a time-accurate manner. The first stage in the time-stepping algorithm is to compute the new momentum components normal to the edges. A local coordinate system (parallel/perpendicular to the cell edges) is used, avoiding the usual decomposition in Cartesian vector components. To achieve this, a new reconstruction algorithm is developed to treat the convection term in the momentum equation. Central or first order upwind differences then suffice. For the computation of the pressure gradient, several schemes have been

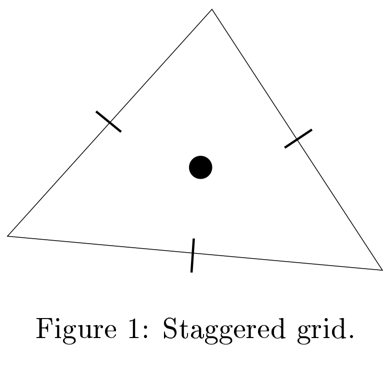

Figure 1: Staggered grid.

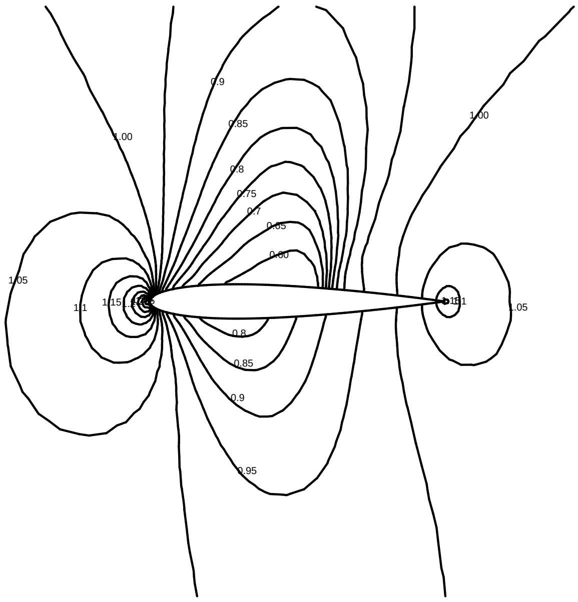

Figure 2: Pressure distribution around a NACA0012 airfoil with $M_\infty = 0.8$ and $\alpha = 1.25^o$.

devised. The second stage is the solution of the continuity equation, resulting in the new density in all cell-centers. Solving the energy equation is the third stage, and the last stage is the computation of the pressure.

Our scheme is much simpler than the prevailing (colocated) schemes for the Euler equations, since only central and/or upwind differences are required, thereby avoiding the necessity to determine numerical fluxes at control volume boundaries by flux-splitting or approximate Riemann solvers, or using solution dependent second and fourth order dissipation terms. The discretization of the continuity and energy equation is straightforward, and the numerical treatment of the convection term in the momentum equation is the only non-trivial issue. Numerical consistency is used to discriminate between the various possibilities that arise.

On the basis of numerical experiments on 1D Riemann problems we have every reason to believe that our scheme approximates genuine weak solutions of the Euler equations that satisfy the entropy condition. A challenging 2D test case is the computation of the transonic flow around a NACA0012 airfoil with an angle of attack of 1.25^o and an inlet Mach number of 0.8. The pressure distribution following from our first order upwind method is shown in Figure 2.