CFD METHODS

COMPRESSIBLE FLOW SIMULATION USING MULTI-DIMENSIONAL UPWIND RESIDUAL DISTRIBUTION SCHEMES

This research is a long term effort towards the development of multidimensional high resolution upwind (MDHR) algorithms for the solution of the compressible Euler and Reynolds-averaged Navier-Stokes equations on unstructured grids. It started in 1986 in collaboration with P.L. Roe and B. van Leer of the University of Michigan and has led to a compact upwind discretization on unstructured grids composed of triangles in 2D, or tetrahedra in 3D. A brief overview of the multidimensional upwind residual distribution method can be found in [MP29, MP31].

To incorporate multidimensional upwinding in a flow solver, the standard finite volume approach using the classical dimensionally split schemes, based on the solution of 1D Riemann problems, is replaced by the residual distribution idea. First, the conservative flux balance (known as the cell residual) for each triangular or tetrahedral cell is computed, based on a linear variation of the unknowns located at the vertices (as in P1 finite elements). Next, this cell-residual is decomposed and distributed in the downwind direction, e.g. the direction of the streamline for the entropy residual. As the multidimensional Euler equations do not completely decouple (except for the entropy equation), matrix distribution schemes have been developed from the earlier scalar schemes [TH3]. The construction of the residual distribution schemes is based on the quasi-linear form of the governing equations. In order to capture discontinuities with the correct jump relations, it is essential that the numerical scheme is consistent with the conservative form of the equations. To ensure this discrete conservation property a multidimensional generalization of the well known 1D Roe linearization is employed.

In recent years the effort concentrated on the development of an efficient implicit solver, based on parallel Newton-Krylov iterative methods, and on the implementation of state-of-the-art turbulence models. The 3D parallel, unstructured grid Euler and Navier-Stokes flow solver *THOR* that was originally developed at the VKI as part of the Ph.D. thesis work of E. van der Weide is currently being improved further in terms of robustness and efficiency.

Despite the many advantages of the residual distribution schemes, they also have some failings in common with the flux difference splitting schemes of dimension by dimension split finite volume methods. In the past year a remedy has been developed for two of the most notable failings.

The first of these failings comes from the fact that the original residual distribution schemes do not satisfy an entropy condition, i.e. they violate the second law of thermodynamics, and therefore allow unphysical expansion shocks to exist in the numerical solution. The entropy fix developed to cure this failing largely follows the ideas of Harten's entropy fix for 1D Riemann solvers and is also based on the observation that expansion shocks are always captured in a single row of cells of the grid. A small amount of artificial dissipation is introduced in these cells such that the expansion shock decays to a gradual expansion and hence the entropy condition is satisfied.

As an example the Mach 3.0 inviscid flow over a forward facing step in a 2D channel is considered. Fig.1 shows the steady state solution computed with the monotone second order upwind scheme on a grid with 17300 nodes. An expansion shock is seen to emanate from the corner over which the flow normally expands gradually. By adding the entropy fix developed for the residual distribution schemes the expansion shock is completely removed as shown in Fig. 2.

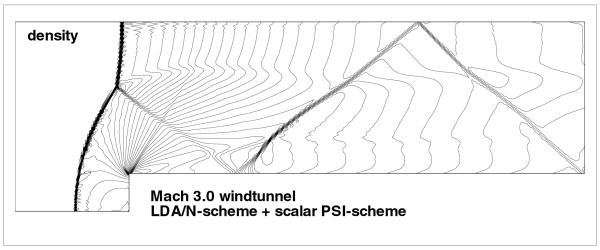

Figure 1: Mach 3.0 flow over forward facing step in a channel. Expansion shock on the corner

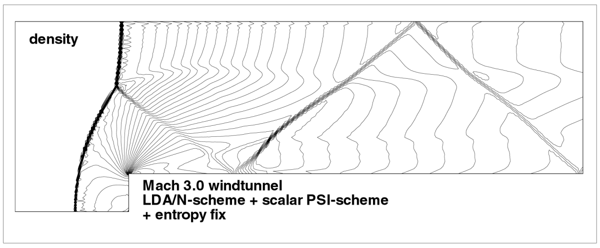

Figure 2: Mach 3.0 flow over forward facing step in a channel. Computed with the entropy-fix added

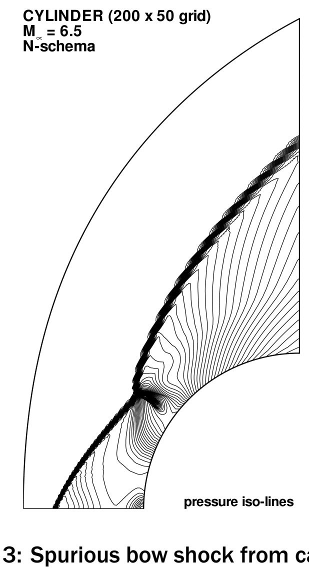

Figure 3: Spurious bow shock from carbuncle phenomenon

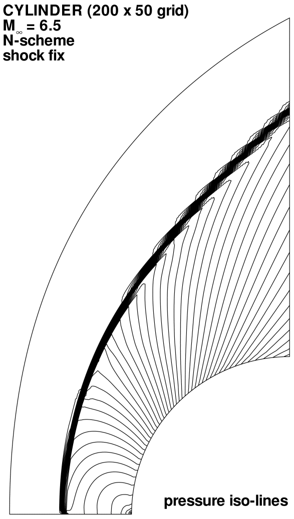

Figure 4: Bow shock computed with shock-fix added

The second failing for which a remedy has been developed is the so called 'carbuncle phenomenon' that may occur in the numerical solution of flows with strong bow shocks. It is a spurious quasi-steady solution which results from a lack of numerical dissipation tangentially to the strong bow shock. As an example, Fig. 3 shows the pressure field of the Mach 6.5 inviscid flow over a cylinder computed with the N scheme (the multidimensional first order upwind scheme) on a 200 × 50 grid. The carbuncle protruding from the bow shock is apparent and it is clear that it completely spoils the solution. To cure this failing a shock-fix was developed that detects strong shocks and adds a small anisotropic diffusion term. The small artificial diffusion added tangentially to the shock is sufficient to avoid the occurrence of the shock instability leading to the carbuncle phenomenon. Fig. 4 shows the effect of adding the shock-fix on the solution of the Mach 6.5 flow over a cylinder.

Since January 1998 the present work on residual distribution schemes and the THOR code also form the basis of the IDeMAS project, a European Union BRITE/EURAM project in aeronautics, co-ordinated by the VKI. The other partners in IDeMAS are INRIA (France), CRS4 (Italy), EPFL (Switzerland), together with major companies in the European aerospace industry (Dassault-Aviation, DASA and Alenia). One of the objectives of the IDeMAS project is to demonstrate for critical aeronautical applications the improved accuracy and efficiency of the present technology compared to standard finite volume methods.

For inviscid flow simulations this objective has been achieved as was demonstrated for the case of the transonic flow over an ONERA M6 wing at $M_\infty = 0.84$ and 3.06° angle of attack, computed on a grid with about 320,000 nodes of which 25,000 are on the wing surface. Fig. 5 shows the iso-Mach lines from the solution computed with the second order monotone multidimensional upwind scheme.

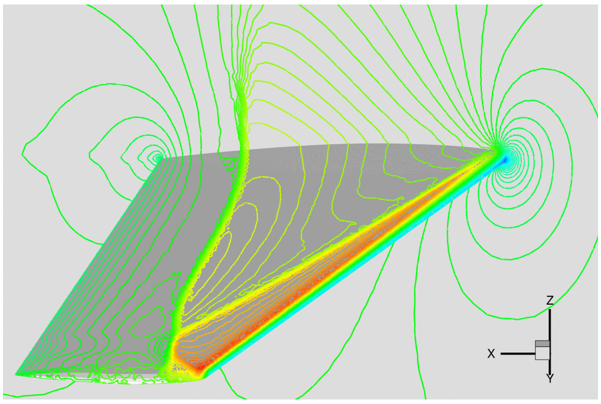

Figure 5: Transonic flow over ONERA M6 wing, M_∞ = 0.84, 3.06°. Iso-Mach lines on the surface and in the symmetry plane

SPACE-TIME RESIDUAL DISTRIBUTION SCHEMES FOR SOLVING HYPERBOLIC CONSERVATION LAWS ON UNSTRUCTURED GRIDS

This project deals with the development of algorithms for the time-accurate solution of hyperbolic conservation laws, with application to the solution of the unsteady Euler equations of compressible gas dynamics.

Over the last decade, a class of multidimensional upwind finite element *space* discretization schemes for solving hyperbolic systems on triangles (2D) and tetrahedra (3D) has been developed at the VKI [MP19, MP31]. The main advantage of the method is that the schemes preserve second order spatial accuracy on the compact stencil of the nearest neighbours for arbitrary geometries. At the same time they have built-in monotonicity and positivity properties allowing excellent discontinuity capturing. However, a strong drawback of these schemes is that it is difficult to maintain these properties if second order accuracy in time is to be obtained.

Recently, R. Abgrall (U. Bordeaux) proposed to use a space-time formulation of the residual distribution schemes. In two space dimensions, the schemes of Abgrall are based on prismatic elements which are triangular in space and linear in time and the solution is approximated by using *bilinear* interpolation over these elements.

In the present work we elaborate on the same ideas, but we use the *standard* distribution schemes developed in the past for spatial triangles and tetrahedra, and apply them to solve *unsteady* problems on space-time geometries. The upwinding property of these schemes allows to decouple the solution on the entire space time domain in a sequence of solutions on space time slabs, consisting of one or two layers of cells in the temporal direction.[1] [2]

It turns out that this decoupling can only be obtained if the space-time mesh satisfies certain properties, and if the timestep for the first layer is limited by a CFL-like condition. However, since no past shielding condition is needed for the second layer, arbitrary CFL numbers

[1] CSÍK, Á.; DECONINCK, H.: *Space Time Residual Distribution Schemes for Hyperbolic Conservation Laws on Linear Unstructured Finite Elements, submitted to the ICFD Conference on Numerical Methods for Fluid Dynamics, Oxford, 2001*

[2] CSÍK, Á.; DECONINCK, H.; POEDTS, S.: *Space Time Residual Distribution Schemes for Hyperbolic Conservation Laws, submitted to the CFD AIAA Conference, Los Angeles, 2001*

can be applied, while maintaining second order accuracy and monotonicity in space time.

Three test cases are discussed below demonstrating the properties of the space time residual distribution schemes on linear elements.

1D oscillating Riemann problem

The first test case is the classical 1D oscillating Riemann problem first proposed by Shu and Osher. The left state is uniform, given by: $\rho = 3.857143$, $\mathrm{v} = 2.629367$, and $p = 10.33333$. In the right state the pressure and the velocity are uniform, $p = 1$, $\mathrm{v} = 0$, while the density is given by $\rho = 1 + 0.2sin(5\pi x)$. In Fig. 1 the squares correspond to the numerical results at $t = 1.8$, obtained by the nonlinear monotone second order B-scheme on a mesh of 401 nodes in space. The solid line corresponds to a solution computed on a mesh containing 1600 nodes. The solution compares favorably with literature results.

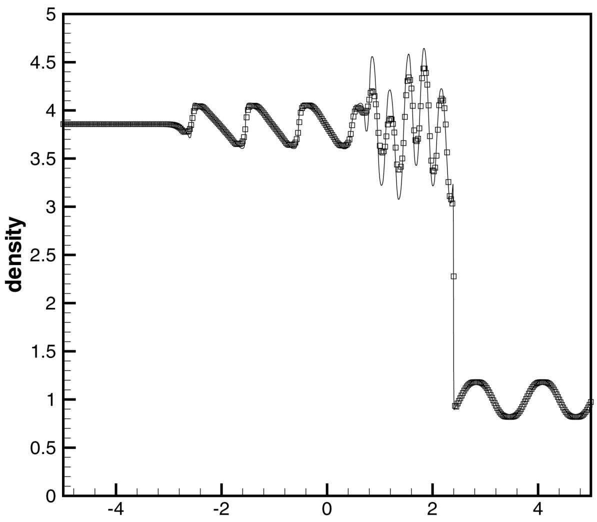

Figure 1: Solution of the 1D oscillating Riemann problem by the second order B scheme at t=1.8

2D sound wave interaction

The second test case concerns the interaction of sound waves in two spatial dimensions. As initial conditions we impose two exponentially decaying pressure perturbations with a maximum amplitude of $\delta p = 0.1$, superposed on a static background with $\rho = 140$, $\mathrm{v} = 0$, and $p = 100$ (Fig. 2,top left). In the solution of this problem the two pressure perturbations propagate outward in the radial direction as linear sound waves, with the speed of sound. On Fig. 2 we show a series of snapshots at different time steps, computed with the LDA scheme on a grid containing 100×100 meshpoints in space. The two sound waves propagate through each other producing interference. This test case illustrates the capability of the method to work well for a stagnant flow.

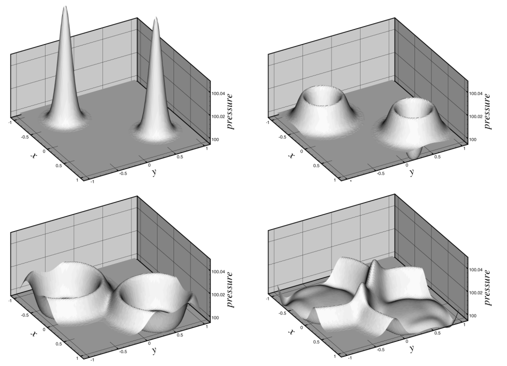

Figure 2: Propagation of linear sound waves. Top left: the initial state at $t = 0$, top right: the solution at $t = 0.25$, bottom left: the solution at $t = 0.5$, bottom right: the solution at $t = 0.75$

2D Riemann problem

We present the solution of a 2D Riemann problem on a mesh containing 100×100 nodes. The initial state of the simulation consists of a square shaped domain with $\rho = 3$, $u = 0$, $v = 0$ and $p = 3$ embedded into an infinite uniform domain with $\rho = 1$, $u = 0$, $v = 0$ and $p = 1$. Due to symmetry reasons we compute the solution over one quarter of the full domain only. On figure 3 and 4 we show the solution at $t = 0.4$ computed by the second order nonlinear monotone B scheme, compared with the first order linear monotone N scheme. On the coordinate axes the solution is identical to the solution of the 1D Riemann problem with the same initial data. However, a complex two-dimensional interaction is seen in the corner region.

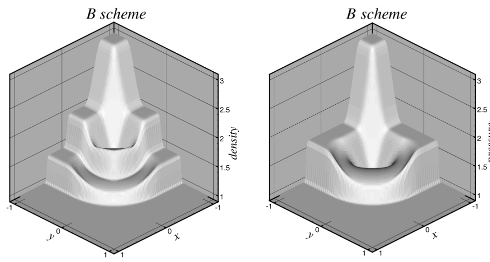

Figure 3: Solution of the 2D Riemann problem by the second order B scheme at $t = 0.4$

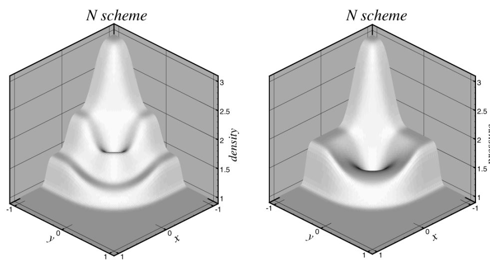

Figure 4: Solution of the 2D Riemann problem by the first order N scheme at $t = 0.4$

HYBRID UNSTRUCTURED GRID GENERATION FOR CONVECTION-DOMINATED FLOW SIMULATIONS

Research on unstructured hybrid grid generation has continued in the framework of the European Union ESPRIT project AMeGOS (Automatic Mesh Generation and Optimisation for industrial flow Simulations). The main task of VKI in this project is the development of a grid generator suitable for convection dominated flow problems with very thin diffusion layers near the boundaries. The main application areas targeted in the project are automotive industry (flow in combustion engines), plating industry (electrochemically reacting flows with mass transfer at the electrode boundaries) and aerospace (high Reynolds number flows). The grid generation package is developed in close collaboration with the software company ElsyCA, which is responsible for Graphical User Interfacing (GUI), CAD interfacing, visualization and interfacing with solvers. Other partners in the projects are AVL, PSA, and Volvo.

The hybrid grid generation approach combines the strengths of structured grid generators to produce stretched anisotropic layers of elements near the boundary with the capacities of unstructured grid methods to cope with complex geometries. The procedure is hierarchic, beginning with the construction of the edge grids which constitute the boundaries of the faces. In a second step the face grids are generated starting from the bounding edge grids, and finally in a third step the volume grids are generated starting from the bounding face grids.

A strong emphasis in the project is given to efficient interfacing with the CAD system. The geometry and topology of the configuration can be taken directly from the output of the CAD system, provided that the CAD definition is exported under the form of a STEP file. The STEP format is a recently developed ISO standard for the electronic exchange of product data, supported by most CAD packages.

In the first year of the project, VKI and ElsyCA have developed in close collaboration the STEP reader, a program module allowing the grid generator to read the topology and geometry of the model configuration (e.g. an airplane or electrochemical reactor).

Further, VKI has developed a hybrid surface grid generator for arbitrary three-dimensional configurations, thus completing the second step in the above algorithm. The requirements on the grid are specified by Grid Control Language (GCL) commands, a STEP-like specification language for grid generation designed in this project. It allows to construct either isotropic or boundary layer grids for which the spacing parameters (e.g. stretching, growth rate) can be controlled by the user for each topological entity (vertex, edge, surface or brep).

Fig. 1 shows an isotropic surface grid for a generic fighter aircraft defined by a STEP file with 191 face entities with geometry described as Coon's patches.

Over the last year, the work has concentrated on isotropic volume meshing. Two different point placement algorithms have been implemented, while a Delaunay triangulation method is used to generate the tetrahedra from a given point cloud.

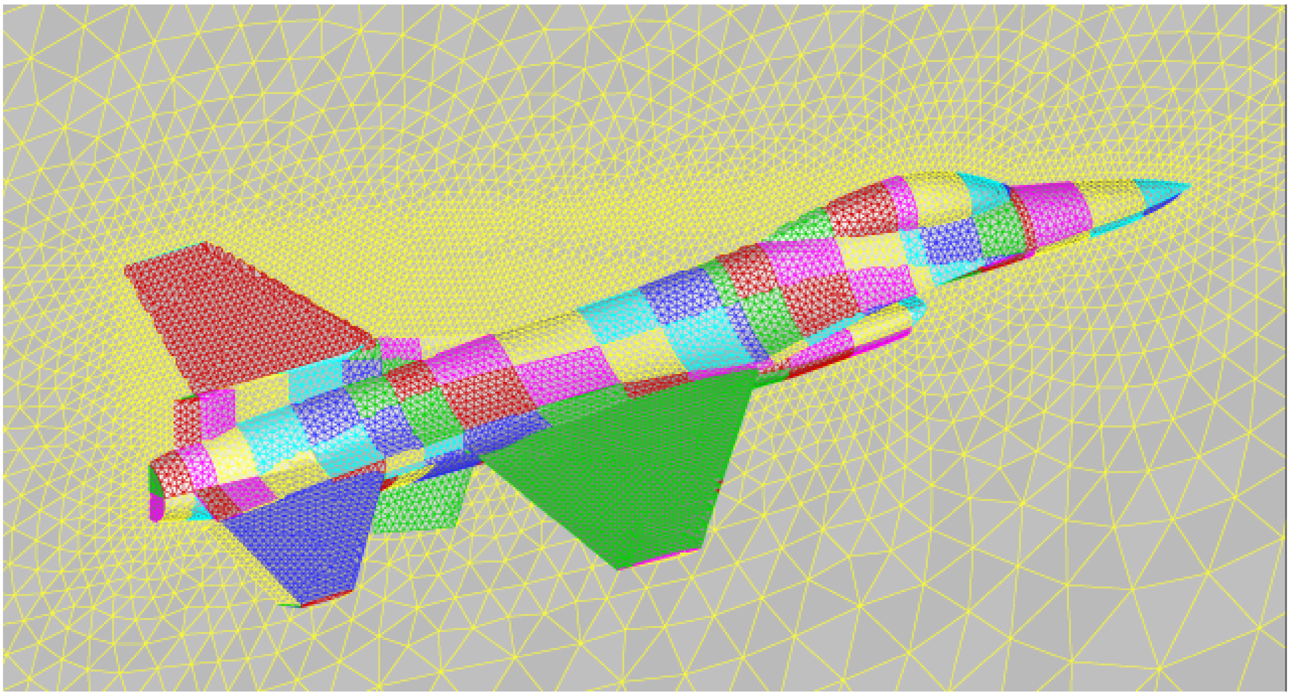

Figure 1: Surface grid for fighter aircraft. Each coloured patch represents a face entity on the STEP file