N96-18082

S11-34
7333
p.9

UNSTRUCTURED VISCOUS FLOW SOLUTION USING ADAPTIVE HYBRID GRIDS

Martin Galle
DLR Institute of Design Aerodynamics
38108 Braunschweig, Germany

SUMMARY

A three dimensional finite volume scheme based on hybrid grids containing both tetrahedral and hexahedral cells is presented. The application to hybrid grids offers the possibility to combine the flexibility of tetrahedral meshes with the accuracy of hexahedral grids. An algorithm to compute a dual mesh for the entire computational domain was developed. The dual mesh technique guarantees conservation in the whole flow field even at interfaces between hexahedral and tetrahedral domains and enables the employment of an accurate upwind flow solver. The hybrid mesh can be adapted to the solution by dividing cells in areas of insufficient resolution. The method is tested on different viscous and inviscid cases for hypersonic, transsonic and subsonic flows.

INTRODUCTION

The development of adaptive methods for efficient and accurate flow calculations has led to two major strategies: One class of schemes is based on the application of triangles in two dimensions and tetrahedral cells in three dimensions. The employment of those grids for adaptive methods yields some advantages: For inviscid calculations the generation of suitable grids even for complex geometries and configurations can be done almost automatically by a powerful grid generator. Furthermore, grid refinement by introducing new points and retriangulating them can be done by some modules of existing grid generators. Such refined grids in general do not require any special treatment either in the metrical setup or in the flow solver part of existing codes. Since the grid generation for complex configurations has become more and more time consuming in comparison to flow calculation, the first point has to be considered to be the main advantage of this class of schemes.

Besides those typical unstructured methods, other adaptive schemes based on quadrilateral cells, or hexahedral cells in three dimensions, exist. Schemes applying to a semi-structured data treatment ([1] and [2]) yield only small advantages concerning flexibility when compared with structured methods. But even hexahedral schemes with an totally unstructured data treatment, that allow an arbitrary cell arrangement, can not compare with tetrahedral schemes in this respect. The main advantage of these methods, contrary to tetrahedral schemes, is the higher efficiency and accuracy especially for regions dominated by viscosity such as boundary layers in high Reynolds number flows. Cells of high aspect ratio can be generated without the need of introducing unwanted small angles. Employing the dual mesh technique, control volume faces are orthogonal to the respective edges.

An approach to combine the advantages of both strategies while circumventing their drawbacks is the employment of hybrid grids. The hybrid grids used in the work presented here consist of hexahedral cells in regions near surfaces, where viscous dominated flow can be expected, and of tetrahedral cells at some distance from those regions to connect the hexahedral domains and the outer boundaries.

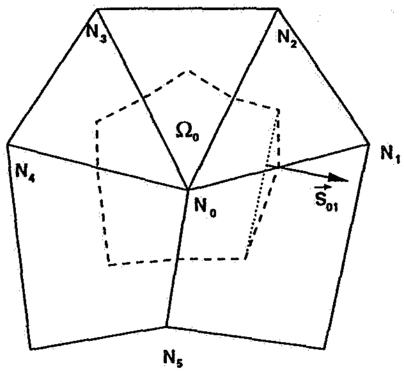

Fig. 1: Dual mesh cell around node N_0

This paper presents an efficient algorithm to solve the Euler- and Navier Stokes equations for hybrid grids consisting of hexahedral and tetrahedral cells. Because of the flexible data structure hexahedral cells do not have to be connected to their neighbors in any predefined arrangement and almost arbitrary combinations of both cell types are allowed. In order to obtain a natural coupling between hexahedral and tetrahedral domains the flow solver part of the presented code applies to a *dual mesh* with control volumes surrounding each node. Fluxes over the control volume boundaries are determined and related to the respective nodes. The dual mesh covers the entire computational domain and connects the two grid types. So, the flow solver does not distinguish between the different cell types, like some comparable methods ([3]). The use of the dual mesh technique yields automatically conservation in the flow field and enables the application of an efficient and accurate upwind flow solver to calculate the inviscid fluxes.

METRICAL SETUP

The input grid data contains information about the geometric positions of the grid nodes and their connections to neighboring nodes. The spatial discretization to determine the viscous and the inviscid fluxes for every node is based on the technique of auxiliary cells used as control volumes. The *metrical setup* encloses the determination of size and shape of the dual mesh cells from the given information.

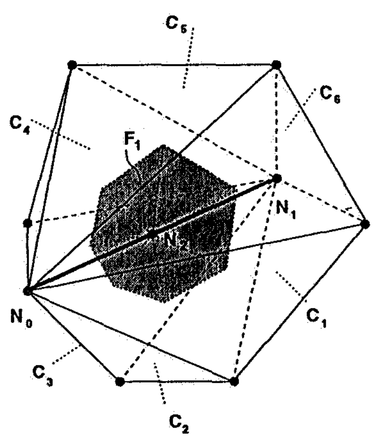

Fig. 2: Face of the dual mesh related to an edge connecting N_0 and N_1

As shown in figure 1, a control volume surrounding node N_0 inside the computational domain has one face for every neighboring node $N_{1..5}$. These faces cross the edges halfway between N_0 and its neighbors and are bounded by the geometric centers of the cells surrounding N_0. The faces of the control volumes are described by face vectors $\vec{S}$, representing the face orientation and the face size. Each edge of the computational grid has one related face in the dual mesh. The face is bounded by the midpoints of the cells surrounding the edge. Because of the need to deal with hanging nodes, control volume faces are not to be bounded by the midpoints of the faces of computational grid cells as e.g. in [4]. As the relations between the cells also have to be determined in order to connect the midpoints of two neighboring cells, the evaluation of the face vectors for three dimensional grids becomes a non trivial task.

Consider an edge connecting the nodes N_0 and N_1 inside the computational domain. This edge is shared by n cells $C_{1..n}$ as shown in figure 2. The Point N_2 is the midpoint of the edge connecting N_0 and N_1. The first task in the setup is the determination of the cells C_1 to C_n and the ascertainment of their order around the edge.

The face F is composed of triangles formed by midpoints of two neighboring cells and N_2. The respective face vector is the sum of all vectors related to the those triangles. The volume Ω_0 of the dual cell is composed of tetrahedra with the corner points N_0, N_2 and the midpoints of two neighboring cells. In a loop running over the cells C_1 to C_n the contributions to the face vector and cell volume are evaluated.

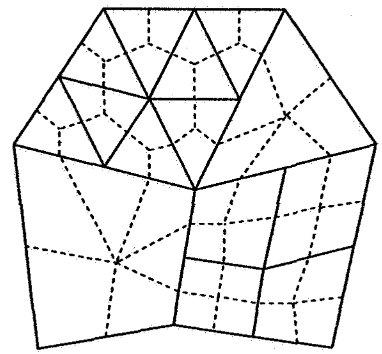

Fig. 3: Dual mesh at the interface between cell types and refinement levels

The metrical setup, including the determination of the volume of the dual mesh cells as well as the components of the face vectors, has to be executed before the flow calculation starts. Since the setup forms one dual mesh from the entire hybrid grid, conservation is guaranteed for the computational domain even at interfaces between regions of hexahedral and tetrahedral cells and at interfaces of refined and unrefined cells, as shown in figure 3.

SPATIAL DISCRETIZATION

Inviscid Flux Calculation

The Euler equations are solved employing a modified AUSM Upwind Scheme ([5]) as it is described in detail by Kroll and Radespiel ([6]). The special merits of AUSM compared to other upwind schemes are the low computational complexity and the low numerical diffusion.

To compute the inviscid flux over the face F is based on the flow conditions on both sides of the face. The values are taken directly from N_0 and N_1 for first order calculations. For second order accurate calculations the independent flow variables are linearly reconstructed on the control volumes around N_0 and N_1 and

$$u_L = u_0 + \nabla u_0 \cdot \frac{1}{2}\vec{V}_{01} \tag{1}$$

The gradient ∇u_0 of a variable u is obtained by employing a Green-Gauss formula:

$$\nabla u_0 = \frac{1}{\Omega_0} \cdot \sum_{i=1}^{n} \frac{1}{2} \cdot (u_0 + u_i) \cdot \vec{S}_{0i} \tag{2}$$

where Ω_0 is the volume of the dual cell around N_0 and $\vec{S}_{0i}$ is the normal vector of the dual mesh face F as shown in figure 1.

Near shocks the values on the edges have to be limited to avoid overshoots. The limiting is done by an minimum/maximum clipping like it is proposed by Barth in [7]. If a reconstructed value at any face of the control volume exceeds the minimum (or maximum) of the values given by node N_0 and the surrounding nodes $N_{1..n}$, the gradient ∇u is scaled by a factor Θ, so the reconstructed value becomes equal to the minimum (or maximum) of the nodes $N_{0..n}$.

Determination of Viscous Flux

In order to calculate the viscous flux over the face of an auxiliary cell the primitive variables and the derivatives of velocities and temperature on the face have to be computed. For Face F in figure 2 these values are evaluated by averaging the values of N_0 and N_1. The gradients of the primitive variables in three cartesian directions are the components of ∇u. For the calculation of the viscous flux the unlimited values have to be used. The determination of the gradients in the way described above provides the gradients in the x-, y- and z-direction. This enables the solution of the full Navier Stokes equations without relying on a Thin-Layer approximation.

GRID ADAPTION

The computational grids can be adapted to the calculated solution by dividing cells in regions of insufficient flow resolution. For tetrahedral cells only an isotropic division into eight children cells is allowed. Hexahedral cells can be divided in one, two or all three directions.

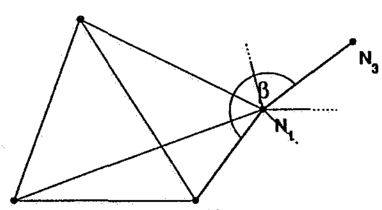

Fig. 4: Determination of helping node N_3

The procedure of tagging cells for division is split into two parts. During the first loop the second differences of pressure Δp with

$$\Delta p = \left| \frac{p_2 - 2p_1 + p_3}{p_2 + 2p_1 + p_3} \right|$$

are computed for each edge. In order to calculate the second differences the most appropriate node N_3 of the neighbors of N_1 has to be determined, as shown in figure 4: N_3 is the node the angle β becomes maximal for. If the value of Δp exceeds a certain threshold, the cell the respective edge belongs to is tagged for division. For hexahedral cells the relative position of the edge in each cell has to be considered in order to divide the cells in the right direction. For the second loop all cells that are already tagged are excluded. During this loop other flow quantities are computed. In the present code the following criteria are implemented:

- first differences of density
- second differences of density
- first differences of velocity

• first differences of velocity weighted by point distance

If the calculated quantity exceeds a second threshold, the respective cells are also tagged for division. The thresholds control the number of divided cells. They can either be fixed before the tagging starts or set iteratively in order to receive a predefined number of cells after the cell division is finished.

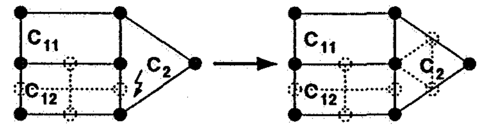

Fig. 5: Tagging of C_2 in order to prevent a second hanging node

For both hexahedral and tetrahedral cells, only one hanging node on every edge is allowed. These hanging nodes are caused by neighboring divided cells as shown in figure 5. The hexahedral cell C_1 is divided once into the children C_{11} and C_{12} and introduces a hanging node on the edge of the tetrahedral cell C_2. If C_{11} was divided again, another hanging node on this edge would be introduced. So C_2 has also to be divided before the next division of C_{11}. An iterative process runs through the field focussing on the tagged cells. If a division would introduce a second hanging node on any edge, the respective cell has also to be tagged for division.

NUMERICAL RESULTS

The first case is the supersonic inviscid flow ($\mathrm{Ma}_\infty = 6$) about a blunt body. A coarse hexahedral mesh was modified in order to obtain a hybrid mesh with hexahedral cells near the body and tetrahedral cells in some distance from the surface. In this test case the metrical setup and the capability of cell division were subject to examinations. Figure 6 shows the position of the shock and the effect of a four times refinement that leads to a good shock resolution even for this coarse initial mesh.

The second case was chosen to test the Navier Stokes formulation of the scheme. For that purpose a pure hexahedral 60x40x3 grid over a flat plate was generated. Figure 7 shows the comparison at different points between the Blasius solution and the computed solution for a subsonic ($\mathrm{Ma}_\infty = 0.5$) laminar flow with a Reynolds Number of 5000. The grid for this case is not adapted to the solution. The results are conform to the analytic solution. Inflow and outflow conditions are computed as proposed by Whitfield in [8].

The first 3D test case is the transsonic flow around an ONERA M6-Wing. Flow conditions are $\mathrm{Ma}_\infty = 0.84$ with an incidence of 3.06°. The grid has been adapted twice. The initial grid contains about 78,000 nodes, 11,000 hexahedra and 370,000 tetrahedra. The once adapted grid contains about 113,000 nodes, 33,000 hexahedra and 403,000 tetrahedra, while the finest grid contains about 211,000 nodes, 95,000 hexahedra and 440,000 tetrahedra. The computed flow field is shown in figure 8. The characteristic λ-shock on the upper side of the wing is nicely resolved. Oscillations at interfaces between cells of different refinement levels occure. Those wiggles are subject to investigations in the future.

Also the AGARD 01 test case has been calculated. Figure 9 shows the five times adapted grid and the respective solution. The shock resolution is acceptable, the shock regions have been refined during each refinement step. The shock on the upper side is located at 0.63 chord length and the shock on the lower side at 0.37 chord length.

The adaptation for the AGARD 03 test case offers more difficulties. As shown in figure 10 the shock behind the airfoil is not resolved very well. The initial grid contains 6,000 nodes. The grid is a three times stacked two dimensional grid. The five times adapted grid contains about 55,000 nodes (also three stacked planes) and the shock distance is 2.75 chord length behind the trailing edge.

CONCLUSIONS

A finite volume scheme using hybrid grids was presented. The employed grids consist of hexahedral cells near body surfaces and tetrahedral cells connecting the hexahedral domains and the outer boundaries. The use of hexahedral cells offers the possibility to resolve viscous dominated flows such as boundary layers efficiently and accurately by applying high aspect ratio cells in those areas. Because of the tetrahedral parts, grids become quite flexible and the generation of grids, even for complex configurations, is relieved very much compared to structured approaches.

In a metrical setup a dual mesh is computed from the initial computational grid. This dual mesh covers the entire computational domain and connects the two grid types naturally. The feasibility of using hybrid grids even for three dimensional flows is shown, but since effective tools for the generation of hybrid grids are not available yet at DLR the presented test cases can not prove the expected advantages of the approach. The calculation of inviscid fluxes is efficient and accurate. Shocks are captured nicely by the employed upwind flow solver. Also the formulation to calculate the viscous fluxes has proved its accuracy. Difficulties still occure at interfaces between refined and unrefined cells.

The next step on the way to an automatic system to compute viscous flows around three dimensional configurations is the extension of the hybrid code to prismatic cells in the vicinity of surfaces. The prismatic cells can substitute for hexahedral cells, as they yield the same advantages as hexahedral cells and algorithms for generating prismatic grids are known from the literature ([9], [10]). The data structure of the new version is changed from a point based to an edge based structure. In order to enable a vectorization a edge colouring is employed. The performance on a NEC-SX 3 is about 1 GFlop, including the metrical setup that is not vectorizable. Also the adaption criteria have to be improved for the new version. This improvement requires intensive studies of different methods.

REFERENCES

[1] M.J. Aftosmis. Viscous flow simulation using an upwind method for hexahedral based adaptive meshes. AIAA 93-0772, 1993.

[2] J.E. Melton, S.D. Thomas, and G. Cappucccio. Unstructured Euler flow solution using hexahedral cell refinement. AIAA-91-0637, 1991.

[3] M. Şoetrisno, S.T. Imlay, and D.W. Roberts. A zonal implicit procedure for hybrid structured-unstructured grids. AIAA-94-0645, 1994.

[4] P. Vijayan and Y. Kallinderis. A 3d finite volume scheme for the Euler equations on adaptive tetrahedral grids. *Journal of Computational Physics*, 113:249–267, 1994.

[5] M.S. Liou and C.J. Steffen. A new flux splitting scheme. *Computers and Fluids*, 107(1):23–39, 1993.

[6] N. Kroll and R. Radespiel. An improved flux vector split discretisation scheme for viscous flows. DLR-FB 93-53, 1993.

[7] T.J. Barth and D.C. Jesperson. The design and application of upwind schemes on unstructured meshes. AIAA-89-0366, 1989.

[8] D.L. Whitfield. Three-dimensional unsteady Euler equation solutions using flux vector splitting, 1983.

[9] K. Nakahashi. Adaptive prismatic grid method for external viscous flow computations. AIAA-93-3314-CP, 1993.

[10] M.J. Marchant and N.P. Weatherill. Unstructured grid generation for viscous flow simulations. In N.P. Weatherill, P.R. Eiseman, J. Häuser, and J.F. Thompson, editors, *Numerical grid generation in computational fluid dynamics and related fields*, pages 151–162, 1994.

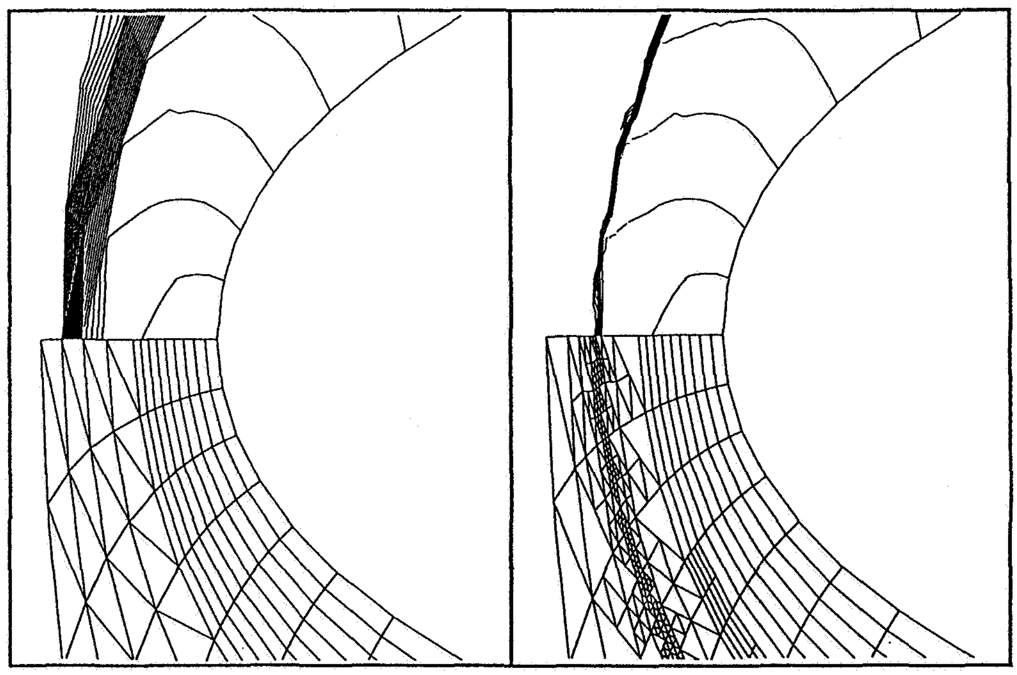

Fig. 6: Flow around a blunt body: Initial grid and four times adapted grid with respective solutions

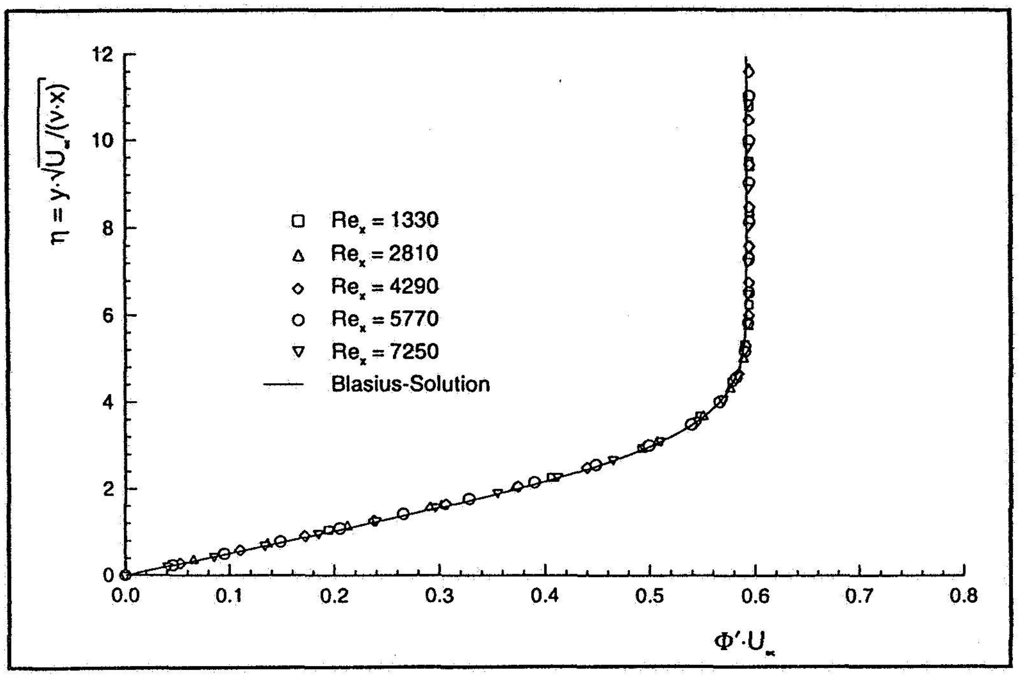

Fig. 7: Laminar flow over a flat plate: Computed Solution compared to analytic Solution

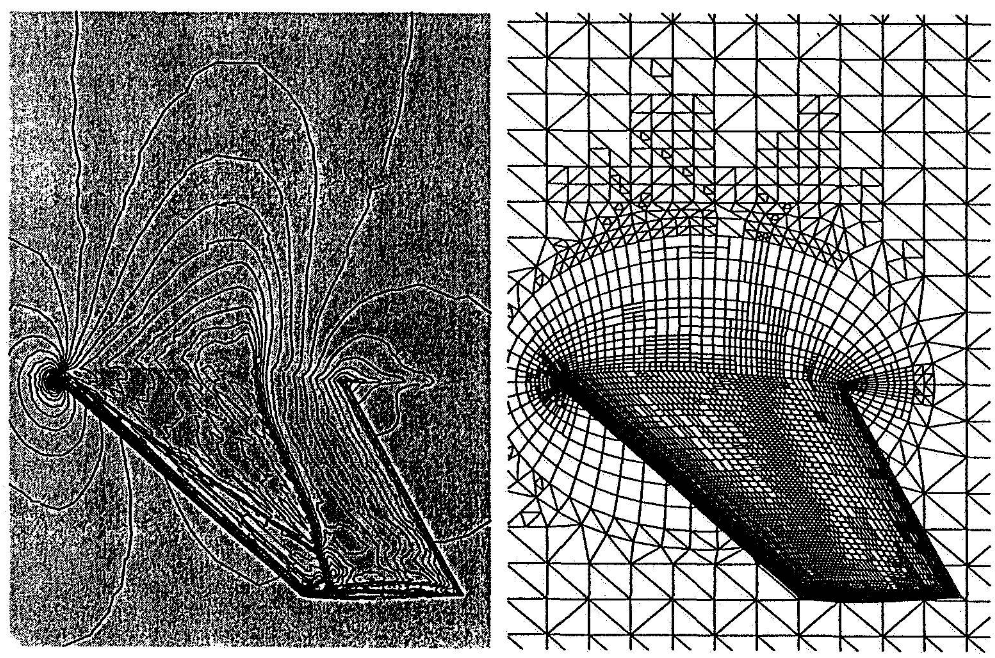

Fig. 8: Transsonic flow around ONERA M6-Wing: twice adapted grid and respective solution

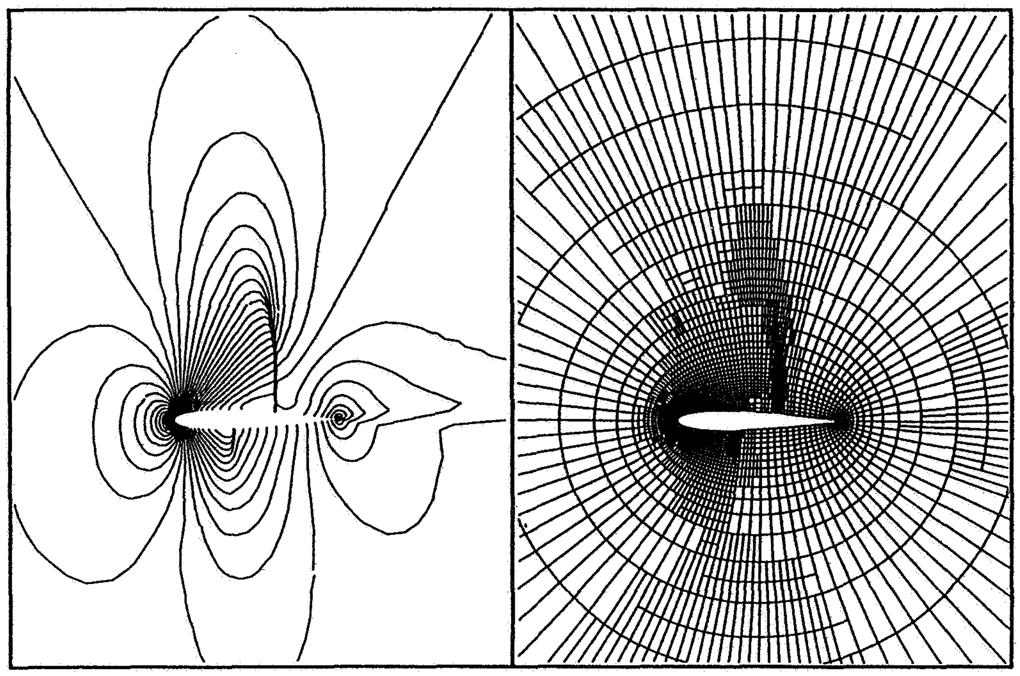

Fig. 9: AGARD01 test case: Five times adapted grid and respective solution

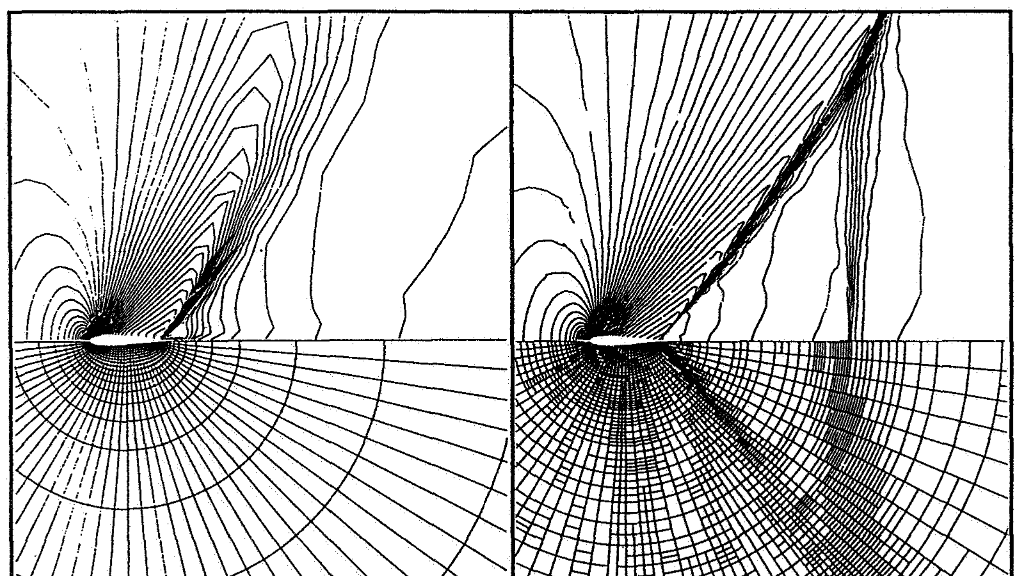

Fig. 10: AGARD03 test case: Initial grid and initial solution and five times adapted grid with respective solution