COMPRESSIBLE TURBULENT FLOW SIMULATION WITH A MULTIGRID MULTIBLOCK METHOD

Hans Kuerten and Bernard Geurts
Department of Applied Mathematics
University of Twente
P.O. Box 217, 7500 AE Enschede, The Netherlands

SUMMARY

We describe a multigrid multiblock method for compressible turbulent flow simulations and present results obtained from calculations on a two-element airfoil. A vertex-based spatial discretization method and explicit multistage Runge-Kutta time-stepping are used. The slow convergence of a single grid method makes the multigrid method, which yields a speed up with a factor of about 20, indispensable. The numerical predictions are in good agreement with experimental results. It is shown that the convergence of the multigrid process depends considerably on the ordering of the various loops. If the block loop is put inside the stage loop the process converges more rapidly than if the block loop is situated outside the stage loop in case a three-stage Runge-Kutta method is used. If a five-stage scheme is used the process does not converge in the latter block ordering. Finally, the process based on the five-stage method is about 60% more efficient than with the three-stage method, if the block loop is inside the stage loop.

INTRODUCTION

Numerical simulations of turbulent flow in aerodynamic applications are frequently based on the Reynolds-averaged Navier-Stokes equations. One of the relevant problems in aeronautics is the prediction of flow quantities in complicated geometries, such as the multi-element airfoil (see figure 1). The simulation of turbulent flow around such a multi-element airfoil configuration was one of the

Figure 1: Geometry of a two-element airfoil.

applications selected for the compressible flow solver which was developed by our group and NLR

as a part of the Dutch ISNaS project [1]. For this application the use of a single-block, boundary-conforming, structured grid is impossible and one may select either an unstructured grid approach or a block-structured grid approach. Although the former technique has been successfully applied by others [2], we selected the block-structured approach in view of the transparent data structure in the coding, ease of implementation of the turbulence model and a high flexibility with respect to the use of different physical models in different parts of the computational domain.

In a previous paper [3] it has been shown that for laminar and turbulent flow around a single airfoil the introduction of the multiblock structure has no influence on the results, with respect to both the steady-state solution and the convergence rate. Furthermore, invoking the Euler equations instead of the Navier-Stokes equations in blocks outside the boundary layer appeared to have no significant influence on the results. In this paper we describe the application of the multiblock concept to the multi-element airfoil. If the Euler equations are used throughout the computational domain a converged steady-state solution is obtained within a reasonable calculation time. However, if the Reynolds-averaged Navier-Stokes equations are solved in the boundary layers, the rate of convergence is unacceptably low. Therefore, a multigrid technique was implemented in order to accelerate the convergence. The resulting gain in calculation time is close to a factor of 20, and the converged solution is in good agreement with wind-tunnel measurements.

In section 2 the numerical technique, which is based on a combination of a finite volume method with central spatial differencing and a Runge-Kutta explicit time-stepping method, is described. The results, both for inviscid and for viscous simulations are presented in section 3. Finally, in section 4 some conclusions are summarized.

NUMERICAL METHOD

In this section we describe the numerical method used in the flow solver. The two-dimensional, compressible Navier-Stokes equations can be written in integral form as

$$\frac{\partial}{\partial t}\left[\int\int_{\Omega} U dx dy\right] + \int_{\partial\Omega}(F dy - G dx) = 0, \tag{1}$$

where U represents the vector of dependent variables,

$$U = [\rho, \rho u, \rho v, E]^T, \tag{2}$$

with ρ the density, u and v the Cartesian velocity components, and E the total energy density. Further, Ω is an arbitrary part of the two-dimensional space with boundary $\partial\Omega$ and F and G are the Cartesian components of the total flux vector. This flux vector consists of two parts: the non-dissipative or 'convective' part and the dissipative or 'viscous' part, which describes the effects of viscosity and heat conduction, and involves first order spatial derivatives. The Navier-Stokes equations (1) are averaged over a sufficiently large time interval. Due to the nonlinear terms in the convective fluxes, the resulting 'Reynolds-averaged Navier-Stokes' equations involve averages of products of two velocity components. These terms are modeled by a suitable turbulence model. In the present paper

the algebraic Baldwin-Lomax turbulence model, in which the unknown terms are modeled by eddy viscosity terms, is adopted [4].

The discretization of the Navier-Stokes equations follows the method of lines, i.e. the spatial discretization is performed first, and subsequently the resulting set of ordinary differential equations is integrated in time, until the steady state solution is approximated. First the computational domain is divided into blocks and each block is partitioned in quadrilateral cells with the help of a structured, boundary-conforming grid. The variables are stored in the grid points. A finite volume method is used in which the integral form of the Navier-Stokes equations is applied to a control volume Ω, bounded by the dashed lines in figure 2. The convective flux through a boundary of this control volume is

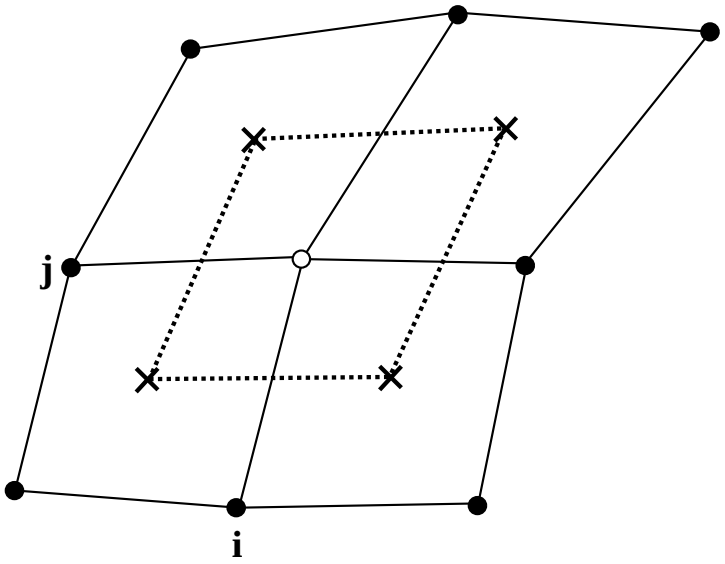

Figure 2: Control volume in the vertex-based method.

approximated using the value of the convective flux vector in the midpoint of the boundary. The latter is calculated by averaging over the two neighboring grid points. The viscous flux vector involves spatial derivatives of the state vector U and is approximated in the corner points of the control volume with the use of Gauss' theorem on a grid cell. The viscous flux is subsequently calculated using the trapezoidal rule. This method is called the vertex-based method.

The method of central differencing leads to a decoupling of odd and even grid points and to oscillations near shock waves. Even in viscous flow calculations the presence of the viscous dissipation is insufficient to damp these instabilities outside shear layers. Therefore, nonlinear artificial dissipation is added to the basic numerical scheme. This artificial dissipation consists of two contributions: fourth order difference terms which prevent odd-even decoupling, and second order difference terms to resolve shock waves. The second order terms are controlled by a shock sensor, which detects discontinuities in the pressure. In the present flow solver the artificial dissipation in the boundary layers, where the viscous dissipation should be dominant, may be reduced by multiplication with the ratio of the local and free-stream Mach number. The role of the artificial dissipation in relation to the viscous

dissipation is discussed in more detail in reference [5].

At the solid wall boundaries the no-slip condition is used. The density and energy density in the grid points on a solid wall are calculated by solving the corresponding discrete conservation laws, using the two adjacent cells within the computational domain and their mirror images inside the wall as control volume. The values of the density and energy density in the grid points inside the walls are adjusted such that the adiabatic wall condition is approximated. The boundary conditions at a (subsonic) far-field boundary are based on characteristic theory. The extent of the computational domain can be reduced without affecting the accuracy if a vortex is superimposed on the incoming free stream outside the computational domain [6].

Due to the topology of the two-element airfoil geometry special points in the computational grid are unavoidable. The computational grids used contain two special points at block boundaries, where five cells meet (see figure 4). These points can be treated in an elegant way within the same numerical scheme, if the dummy vertices outside the 'current' block are defined appropriately. The multi-valuedness of the variables at the special point, caused by this asymmetric treatment, is eliminated by taking the average of the five different values after all blocks have been treated. This is sketched in figure 3.

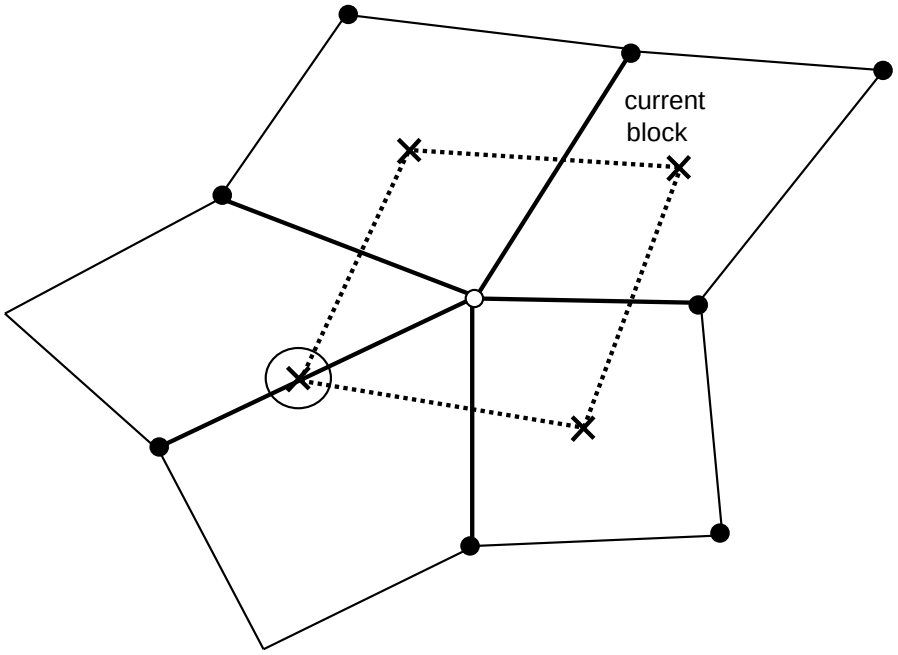

Figure 3: Control volume for a special point.

The system of ordinary differential equations, which results after spatial discretization, is integrated in time using a time-explicit multistage Runge-Kutta method. In the present flow solver a three-stage scheme in which the dissipative fluxes (both viscous and artificial) are calculated once per time-step, and a five-stage scheme in which the dissipative terms are calculated only at the odd stages, are implemented. With this treatment both calculation time is saved and the stability region of the method is increased. Extra calculation time is saved by advancing each grid point at the maximum

local time-step according to its own stability limit. In this way the evolution from the initial solution to the steady state is no longer time accurate, but the steady state solution obtained is unaffected.

The above time-stepping method acts as the relaxation method and coarse grid operator in the multigrid solver (see reference [6]). In this solver an initial solution on the finest grid is obtained with a full multigrid method. This initial solution is corrected in the FAS-stage, where either V- or W-cycles can be chosen. A fixed number of pre- and post-relaxations is performed before turning to the next coarser or finer grid. The solution is transferred to a coarser grid by injection, the residuals by full weighting and the corrections to the solution are prolonged by bilinear interpolation. In order to increase the smoothing properties of the Runge-Kutta time-stepping technique an implicit averaging of the residuals is applied with frozen residuals at the block boundaries. For mono-block applications this method has given satisfactory results for both two-dimensional and three-dimensional flows [5].

In the multi-element airfoil application care has to be taken in the definition of the residual-vector in the special points. The proposed treatment of a special point implies that the control volume is different in each of the five blocks where such a point is found. In the required averaging the five residual-vectors in a special point are weighed with their corresponding time-steps. Without this weighing the multigrid process cannot converge to the single grid stationary state solution.

In this multigrid, multiblock solver with a multistage time-stepping method there are various possibilities for intertwining the different loops. In the present study the grid loop is chosen as the outer loop and the effect of interchanging the block and the stage loop will be studied. Several 'competing' requirements serve as possible guidance for selecting a specific ordering of these loops. On the one hand an anticipated parallel processing of the different blocks is more efficient, if the data transfer between the blocks is kept to a minimum, i.e. with the stage loop inside the block loop. On the other hand the good convergence of the multigrid mono-block solver may be reduced as the dummy variables near the block boundaries are kept frozen during more stages of the time-step. This would suggest to put the block loop inside the stage loop. In order to study this dilemma we implemented these two loop orders in a flexible way: a single parameter determines whether the block loop is situated inside or outside the stage loop.

RESULTS

Description of the test-case

We will present results for a two-component airfoil geometry consisting of the NLR7301 wing section, from which a flap has been cut out at a deflection angle of 20° and with a gap width of 2.6% chord length [7] (see figure 1). The combination of a Mach number of 0.185 and an angle of incidence of 6° or 13.1°, of which the latter is close to maximum lift conditions, yields subsonic flow. The Reynolds number based on the chord length of the airfoil is 2.51×10^6. In the viscous calculations

the locations of the transition from laminar to turbulent flow are prescribed.

The C-type computational grids (either for inviscid or viscous flow) were constructed by J.J. Benton from British Aerospace, and are subdivided in 37 blocks (see figure 4). The grid lines are continuous over block boundaries. Two grids are used: one 'Euler' grid (inviscid) consisting of 16448 cells, and a 'Navier-Stokes' grid (viscous), which is refined in the boundary layers and wakes and consists of 28288 cells.

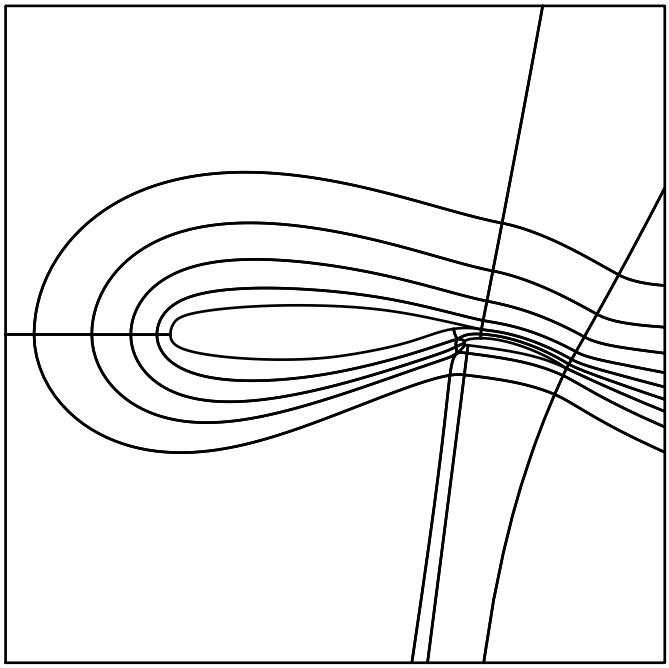

Figure 4: Block structure of the computational grid.

For both angles of incidence results from wind-tunnel measurement by Van den Berg [7] are available, including velocity profiles in the boundary layers and the pressure coefficient on the profile. Since the flow is attached apart from a small laminar separation bubble near the leading edge of the wing, the adopted turbulence model should be adequate and yield a useful comparison between experiment and calculation.

Inviscid Flow

In order to test the flow solver on the complicated block structure of the two-element airfoil geometry, we considered the relatively simple inviscid flow case, where in all blocks the Euler equations are solved. In this way problems related to the turbulence model are separated from possible algorithmic

problems. The use of the Euler equations implies that the boundary conditions at the solid wall boundaries have to be changed. For inviscid flow there is only one physical boundary condition of zero mass flux through the wall. In the vertex based approach the density, the pressure and the tangential velocity at the wall are approximated by linear extrapolation.

In figure 5 the multigrid convergence behavior of the solver in the 13.1° case is shown. The discrete L_2-norm of the residual of the density is plotted as a function of the number of W-cycles. A converged solution is obtained within a much smaller calculation time when compared to the single grid approach even though only three different grid levels are available. Both for the single grid and the multigrid calculations machine accuracy was obtained. The specific block structure nor the treatment of the special points leads to any specific difficulties. For this inviscid test a comparison with experimental results is not meaningful and will not be made.

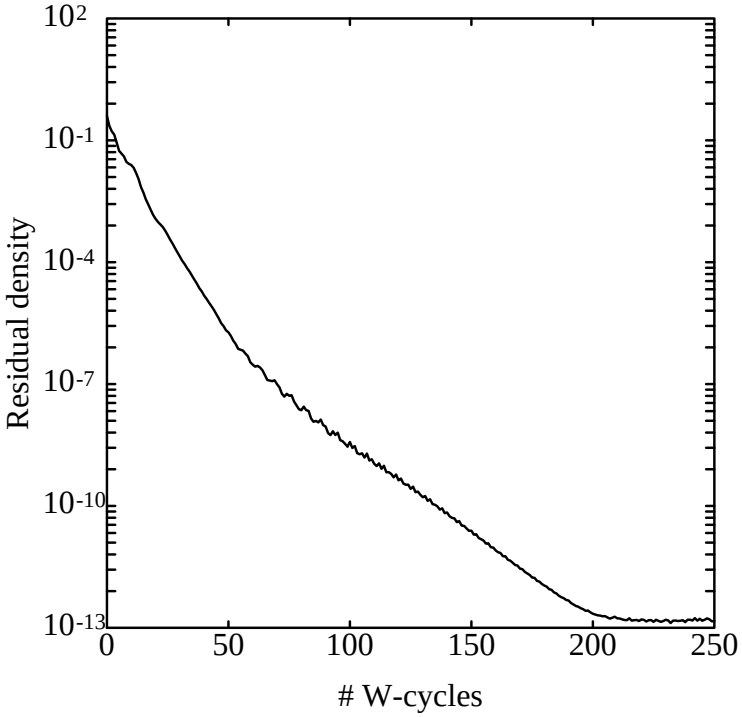

Figure 5: Convergence behavior for inviscid flow at an angle of incidence of 13.1°.

Viscous Flow

We consider the simulations of turbulent, viscous flow and present results for the 6° case only. Single-grid calculations in which only local time-stepping is applied as a convergence acceleration technique yield a steady-state solution which is in good agreement with the experimental results. However, in contrast with a fully inviscid simulation, the rate of convergence is very small, and

renders this method unacceptable for practical applications. Therefore, as a method to increase the convergence rate further, the multigrid technique and implicit residual averaging as described in section 2 are indispensable.

In a simulation of turbulent flow at high Reynolds number it is important that the effects related to the physical dissipation are not outweighed by those of the numerical or artificial dissipation. This requirement could give rise to difficulties in the present multigrid method, since the time-stepping method used requires a certain minimum amount of dissipation for sufficient smoothing of the large wave-number components of the error (see reference [5]). If the artificial dissipation in the boundary layer is reduced by scaling with the ratio of the local and free-stream Mach number, i.e. decreasing the smoothing properties of the time-stepping method, a converged solution (engineering accuracy) could be obtained by increasing the number of pre- and post-relaxations. The convergence behavior of this calculation during the FAS stage is shown in figure 6, where the discrete L_2-norm of the residual of the density is plotted as a function of the number of W-cycles. In the blocks outside the boundary layers and wakes the Euler equations are solved instead of the Navier-Stokes equations. The good

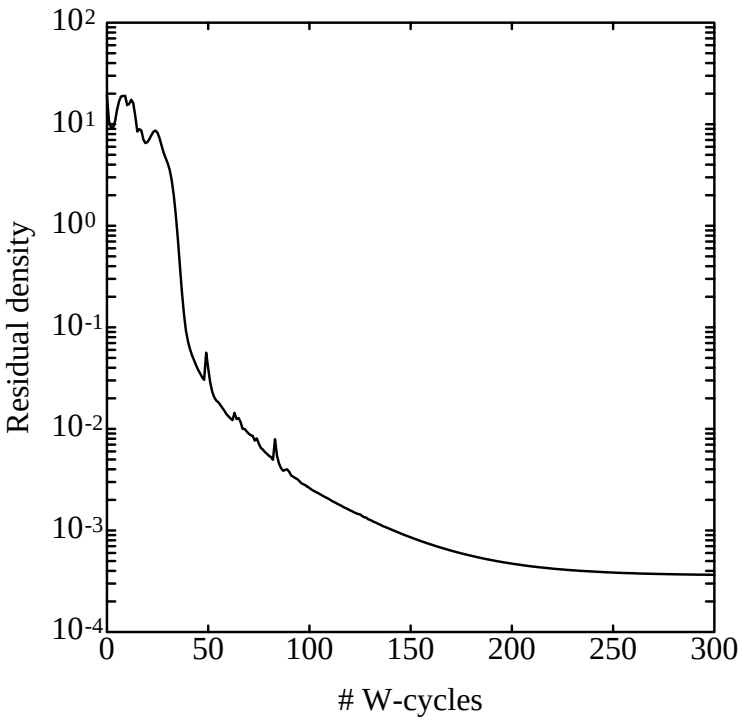

Figure 6: Viscous flow at an angle of incidence of 6.0°: convergence behavior

agreement with the wind-tunnel measurements can be inferred from figure 7, where the experimental and numerically predicted pressure coefficient on the airfoil and flap are shown.

This solution was obtained with the block loop inside the stage loop of the five-stage Runge-Kutta time-stepping method. Hence, the variables at the dummy vertices outside a block are updated after every stage, which implies that the effects of the multiblock structure on the convergence are kept to a minimum. The frequency of data transfer between the blocks makes this method less

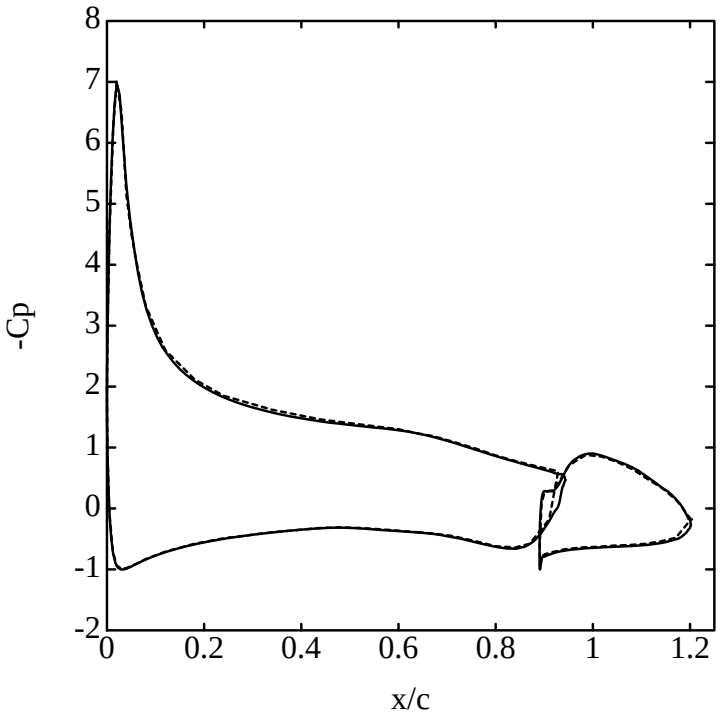

Figure 7: Viscous flow at an angle of incidence of 6.0°: comparison of the pressure coefficient on the airfoil between calculation (solid) and experiment (dashed).

efficient for parallel processing. However, with the block loop outside the stage loop, i.e. with an update of the dummy variables only after five flux evaluations, a converged solution could not be obtained. Apparently, the interval between two moments of data transfer between the blocks has to be sufficiently small in order to obtain a convergent multigrid method.

Further evidence for this statement is obtained from calculations with a three-stage instead of a five-stage Runge-Kutta time-stepping method. If the block loop is outside the stage loop, the dummy variables are updated after three flux evaluations. Although the rate of convergence is lower than in the case with the loops interchanged (see figure 8), the solution has converged within engineering accuracy after $\approx$ 200 W-cycles. A comparison of the three-stage and five-stage schemes with the block loop inside the stage loop shows that the five-stage scheme is more efficient: about 60 W-cycles suffice to get the residuals at the same level as with the three-stage scheme after 200 W-cycles. The five-stage scheme leads to a reduction in calculation time of approximately 60% in this instance.

DISCUSSION

We presented simulation results obtained with a multigrid multiblock method for a two-element airfoil. Both viscous and inviscid calculations were performed using the same multigrid process and the same vertex-based spatial discretization method. Moreover, either a three- or a five-stage

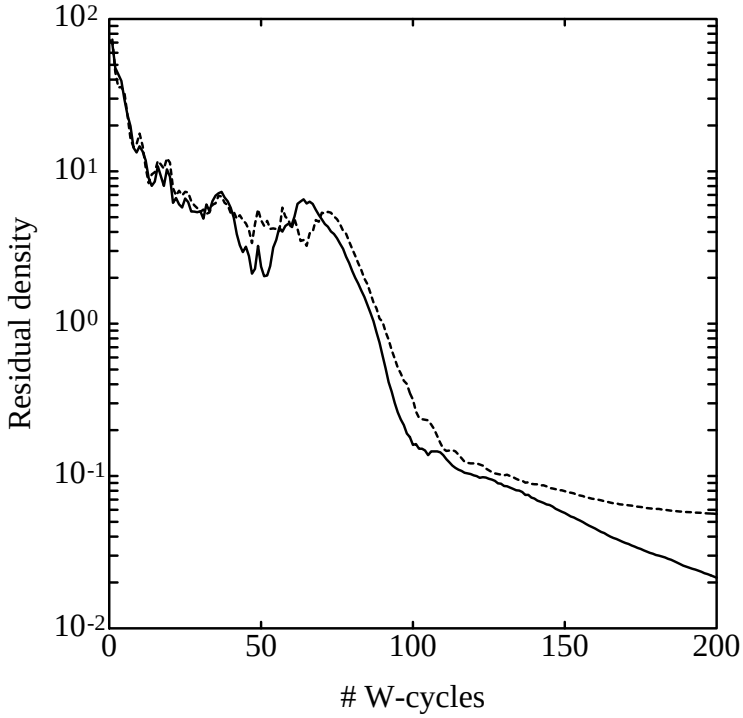

Figure 8: Convergence behavior of the three-stage Runge-Kutta scheme for turbulent flow; comparison between block loop inside (solid) and outside (dashed) stage loop.

Runge-Kutta scheme was considered for the integration in time and the smoothing properties of this relaxation method were further enhanced through the introduction of local time-stepping, implicit residual averaging in which the residuals at the block boundaries were kept fixed to their non-smoothed values.

The inviscid calculations have shown that a solution which is converged up to machine accuracy can be obtained with this multigrid method. A comparison with the single grid simulation method shows that a considerable reduction in calculation time was obtained with the multigrid method, although the convergence of the single grid method for inviscid calculations was already quite acceptable. We also investigated two different numerical boundary conditions at the solid walls. It appeared that linear extrapolation of the pressure not only leads to a better convergence than constant extrapolation, but also gives rise to a much smaller entropy layer around the airfoil. The resulting drag coefficient, which theoretically should equal zero in this subsonic flow, is reduced by almost 60%.

In the viscous calculations the single grid method was found to yield a well converged result in the 6°-case, however, the convergence towards the steady state solution was extremely slow and makes the use of a multigrid approach essential. A comparison of the calculation times required in both methods shows that a total speed-up with a factor of about 20 can be reached. The numerical predictions obtained for the lift- and pressure coefficients compare well with experimental results and give confidence in the use of the Baldwin-Lomax model for this application. The convergence of the multigrid process was studied in detail, showing that the ordering of the various loops in the

process has a considerable effect. Interchanging the block- and stage loops, keeping the grid loop as the outer loop yields an optimal convergence when the block loop is put inside the stage loop. If the stage loop is put outside the block loop then convergence of the multigrid process was absent when using the five-stage Runge-Kutta method as the relaxation method. Apparently, the smoothing of the relaxation method becomes less effective as the number of stages between two 'updates' of the dummy-variables increases. This result has some less favorable consequences in view of a possible parallel processing of the multigrid method. On the one hand parallel processing seems more efficient if the frequency of data transfer between the blocks can be reduced. On the other hand the reduction of this frequency results in a reduction of the convergence rate of the multigrid process, and in some instances even to an absence of convergence. This suggests that in a possible parallel processing of this multigrid method, an optimal rate of data-exchange between the blocks should be determined.

Acknowledgement

The authors are greatly indebted to Frans Brandsma for several useful and stimulating discussions.

References

1. Brandsma, F.J.; Vogels, M.E.S.; Van der Vooren, J.; Dijkstra, D.; and Kuerten, J.G.M.: Predesign document of the ISNaS compressible flow simulator. ISNaS-88.04.027, April 1988.

2. Mavriplis, D.J.: Turbulent Flow Calculations using Unstructured and Adaptive Meshes. in *Proceedings of the 12th International Conference on Numerical Methods in Fluid Dynamics*, Oxford, July 9-13 1990, pp. 228-232.

3. Geurts, B.J.; and Kuerten, H.: Numerical Aspects of a Block Structured Compressible Flow Solver. *J. Engg. Math.*, vol. 27, 1993, pp. 195-214.

4. Baldwin, B.; and Lomax, H.: Thin layer approximation and algebraic model for separated turbulent flow. AIAA-78-257, 1978.

5. Kuerten, J.G.M.; Geurts, B.J.; Van der Burg, J.W.; Vreman, A.W.; and Zandbergen, P.J.: Development and applications of a 3-D compressible Navier-Stokes solver. in *Proceedings of the 13th International Conference on Numerical methods in Fluid Dynamics*, Rome, July 6-10 1992.

6. Radespiel, R.: A cell-vertex multigrid method for the Navier-Stokes equations. NASA-TM-101557, 1989.

7. Van den Berg, B.: Boundary layer measurements on a two-dimensional wing with flap. NLR-TR-79009-U, 1979.

View publication stats