Mecánica Computacional Vol. XVI
compilado por G. Etse y B. Luccioni
Tucumán, Septiembre de 1996

SOLUTION OF INCOMPRESSIBLE AND COMPRESSIBLE FLUID FLOWS BASED ON RUNGE-KUTTA TIME-STEPPING SCHEMES

DE BORTOLI, A. L.
Department of Mechanical Engineering - PROMEC
Federal University of Rio Grande do Sul
Sarmento Leite, 425 - 90050-170 - Porto Alegre - Brazil

ABSTRACT

The great development of numerical calculation lead to a variety of methods, which are different in their solution algorithms as well as in the discretization of the governing equations. Although this development, many work still remains to improve them towards a fast, accurate and stable convergence. This work employs a numerical method to solve compressible and incompressible fluid flows using a finite-volume, explicit Runge-Kutta multistage scheme, with central spatial discretization in combination with multigrid and preconditioning. The discretization used follows the cell- and node-centered arrangement of the control volume and update points for the flow variables. Numerical results are presented for NACA 0012 airfoil and a three dimensional channel for the Euler equations. Mach number is varied between 0.8 and 0.002.

INTRODUCTION

Numerical simulation plays nowadays an important role to predict the flow field in many situations. The rapid evolution of computational fluid dynamics has been driven by the need of faster and most accurate methods for the calculation of flow fields around configurations of technical interest.

Numerical methods allow us to analyse some important phenomena, where experiments are usually too expensive or even impossible. Common methods employed for the solution of fluid flows are the finite differences, finite volume, finite elements and boundary elements. Each of these methods has its advantages and disadvantages.

In this way, the interest is the development of efficient methodologies that can be employed to solve compressible and incompressible fluid flows. Extensions of the numerical methods employed to solve compressible [1] [2] and incompressible [3] [4] fluid flows have been applied, with success, for the solution of all speed flows. However, there are no general methods for all situations. As the used methodologies are limited in what concerns to accuracy, cost and applicability, they must be investigated in order to design at low costs.

Several formulations of the Euler equations are known for different applications of numerical methods. The natural way for the derivation of the Euler equations is to set up the conservation of mass, momentum and energy in integral form, which is the adequate model for non visvous flows. In this way only a minimum of assumptions of the functions are required.

The discretization employed in this work is based on the finite volume explicit Runge-Kutta multistage scheme [5] [6], with central spatial discretization in combination with multigrid and preconditioning [2]. The idea of the multigrid approach is the use of a sequence of successively coarser meshes in combination with local time-stepping and residual averaging to efficiently damp

disturbances of error through the flow field. Results obtained with the Euler equations are validated using the potential (analytical) equation solutions or experimental data found in the literature. Mach number is varied between 0.8 and 0.002.

The standard compressible method is employed for the solution of compressible and incompressible fluid flows. It consists of the use of the momentum equations to calculate the velocity components, the energy equation to calculate the energy, the mass conservation equation to calculate the density and the state equation to calculate the pressure.

In order to eliminate numerical difficulties solving incompressible flows, preconditioning is employed [7] [1]. It consists of the transformation of the governing equations into a form that is numerically better conditioned, ensuring relatively rapid convergence [2]. The preconditioning system has a rate of convergence almost independent from the Mach number. Moreover, the discrete solution is highly improved, eliminating spurious oscillations frequently encountered in incompressible flows [8].

GOVERNING EQUATIONS

The governing equations for non viscous flows are the Euler equations. The three dimensional Euler equations for unsteady incompressible inviscid flows in integral form reads

$$\frac{\partial \vec{W}}{\partial t}+\frac{\partial \vec{F}_1}{\partial x}+\frac{\partial \vec{F}_2}{\partial y}+\frac{\partial \vec{F}_3}{\partial z}=0 \qquad (1)$$

where

$$\vec{W}=\begin{Bmatrix}\rho\\ \rho u\\ \rho v\\ \rho w\\ \rho E\end{Bmatrix},\quad \overline{\overline{F}}=\begin{Bmatrix}\rho\vec{q}\\ \rho u\vec{q}+p.\vec{i}\\ \rho v\vec{q}+p.\vec{j}\\ \rho w\vec{q}+p.\vec{k}\\ \rho H\vec{q}\end{Bmatrix},\qquad \overline{\overline{F}}=\vec{F}_1\vec{i}+\vec{F}_2\vec{j}+\vec{F}_3\vec{k}$$

where ρ is the fluid density, $\vec{q}$ the velocity vector ($\vec{q}=u\vec{i}+v\vec{j}+w\vec{k}$) and p the pressure.
Since the integral form allow discontinuities, the approach is suitable for capturing shocks and discontinuities in the flow field. The total energy E and total enthalpy H are

$$E=e+\frac{u^2+v^2+w^2}{2},\qquad H=E+\frac{p}{\rho}$$

To close this system of equations the state equation for a perfect gas is employed [9]

$$p=\rho RT=(\gamma-1)\rho[E-\frac{u^2+v^2+w^2}{2}]$$

where R is the universal gas constant and γ the specific heat ratio. Eq. (1) can be cast into the integral form [6]

$$\int_V \frac{\partial \vec{W}}{\partial t}dV+\int_S(\overline{\overline{F}}.\vec{n})dS=0$$

It is well known that when the magnitude of velocity becomes small, in comparison with the acoustic speed, the time-marching schemes converge very slowly. Therefore preconditioning is employed. It consists basically in multiplying the vector $\vec{W}$ by a special matrix, which modifies the form of the

governing equations. Based on the conservative variables, the following preconditioning matrix is employed:

$$\Gamma\frac{\partial\vec{W}}{\partial t}+\frac{\partial\vec{F}_1}{\partial x}+\frac{\partial\vec{F}_2}{\partial y}+\frac{\partial\vec{F}_3}{\partial z}=0$$

where [7] [1]

$$\Gamma=\begin{bmatrix} 1 & 0 & 0 & 0 & 0 \\ 0 & 1 & 0 & 0 & 0 \\ 0 & 0 & 1 & 0 & 0 \\ 0 & 0 & 0 & 1 & 0 \\ \frac{(u^2+v^2+w^2)}{2}(M^{-2}-1) & u(1-M^{-2}) & v(1-M^{-2}) & w(1-M^{-2}) & M^{-2} \end{bmatrix}$$

Inspection of this preconditioning matrix indicates that the energy equation is transformed into temperature for low Mach-numbers. Thus, the eigenvalues of the resultant system of equations will be very similar when the Mach-number goes to zero, laying the basis of construction of efficient solvers [1] to solve incompressible flows [2].

DESCRIPTION OF THE NUMERICAL METHOD

One of the differences among the various finite volume formulations known in the literature is the arrangement of the control volume and update points for the flow variables [5] [8]. The most frequently used schemes are the cell-centered, cell-vertex and node-centered approach. Each of these schemes has its advantages and disadvantages. The discretization used is based on the cell- and node-centered arrangements [10].

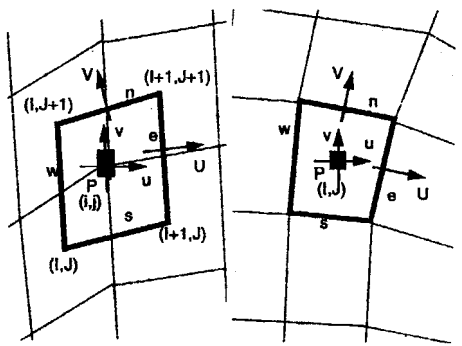

Fig. 1 Node-centered and cell-centered arrangements (bidimensional)

In the computational domain the cell vertices are identified by their indices (I,J,K). As Eq. (1) is valid for arbitrary control volume, it is also valid for $V_{i,j,k}$, that means

$$\frac{\partial\vec{W}_{i,j,k}}{\partial t}=-\frac{1}{V_{i,j,k}}\int_S(\overline{\overline{F}}.\vec{n})dS$$

The finite volume discretization based on the central averaging is not dissipative [6]. The numerical procedure does not converge to the steady state solution when the high frequency oscillations of

error in the solution are not damped. In order to avoid these oscillations dissipative terms $\vec{D}_{i,j,k}$ have to be introduced as follows [6]

$$\frac{\partial \vec{W}_{i,j,k}}{\partial t}+\frac{1}{V_{i,j,k}}[\vec{Q}_{i,j,k}-\vec{D}_{i,j,k}]=0$$

This dissipation operator is a blend of second and fourth differences and is defined according to [11]

$$\vec{D}_{i,j,k}=d_{i+1/2,j,k}-d_{i-1/2,j,k}+d_{i,j+1/2,k}-d_{i,j-1/2,k}+d_{i,j,k+1/2}-d_{i,j,k-1/2}$$

whose dissipation coefficient is given by

$$d_{i+1/2,j,k}=\alpha_{i+1/2,j,k}\{\varepsilon^{(2)}_{i+1/2,j,k}\delta_x\vec{W}_{i,j,k}-\varepsilon^{(4)}_{i+1/2,j,k}\delta_{xxx}\vec{W}_{i-1,j,k}\}$$

The difference operators of first and third order are δ_x and δ_{xxx}, respectively, and α is the scaling factor, which is written for the i direction as [10]

$$\alpha_{i+1/2,j,k}=\frac{1}{2}(\lambda^{*}_{i,j,k}+\lambda^{*}_{i+1,j,k})$$

where

$$\lambda^{*}_{i,j,k}=\lambda^{i}_{i,j,k}\phi^{i}_{i,j,k},$$

$$\phi^{i}_{i,j,k}=1+max\left[\left(\frac{\lambda^{j}_{i,j,k}}{\lambda^{i}_{i,j,k}}\right)^{w},\left(\frac{\lambda^{k}_{i,j,k}}{\lambda^{i}_{i,j,k}}\right)^{w}\right].$$

The coefficients adapted to the local pressure gradients $\varepsilon^{(2)}$ and $\varepsilon^{(4)}$, needed to obtain the dissipation coefficient, are written as

$$\varepsilon^{(2)}_{i+1/2,j,k}=k^{(2)}max(\upsilon_{max}),$$

$$\varepsilon^{(4)}_{i+1/2,j,k}=max(0,k^{(4)}-\varepsilon^{(2)}_{i+1/2,j,k}),$$

where

$$\upsilon_{i,j,k}=\frac{|p_{i+1,j,k}-2p_{i,j,k}+p_{i-1,j,k}|}{|p_{i+1,j,k}+2p_{i,j,k}+p_{i-1,j,k}|},$$

$$\upsilon_{max}=(\upsilon_{i+2,j,k},\upsilon_{i+1,j,k},\upsilon_{i,j,k},\upsilon_{i-1,j,k}),$$

and

$$0.5\le k^{(2)}\le 0.6,\qquad \frac{1}{128}\le k^{(4)}\le\frac{1}{48}.$$

The spectral radius λ is defined for the i direction according to [7]

$$\lambda^{i}=\frac{u(1+M^2)+\sqrt{u^2(1-M^2)^2+\beta^2c^2}}{2}$$

It is well known, that for a central difference scheme zero artificial dissipation viscosity creates numerical difficulties. Therefore β^2 is chosen as [2]

$$\beta^2 = max(4M^2, \varphi),$$

where

$$0.1 \le \varphi \le 0.6$$

Finally, the preconditioning is applied only for the convective terms as follows

$$\frac{\partial \vec{W}_{i,j,k}}{\partial} + \frac{1}{V_{i,j,k}}[\Gamma^{-1}\vec{Q}_{i,j,k} - \vec{D}_{i,j,k}] = 0$$

TIME-STEPPING AND ACCELERATION TECHNIQUES

In order to obtain numerical solutions of high accuracy, the Runge-Kutta method is chosen [5]. This method is characterised by its low operation count. More than two stages are employed in order to extend the stability region. The classical fourth order Runge-Kutta method requires the evaluation of many coefficients and dissipative terms, what leads to storage problems. Therefore, the following multistage scheme, which requires few computational storage, is employed [6]

$$\vec{W}^{(0)}_{i,j,k} = \vec{W}^{n}_{i,j,k}$$

$$\vec{W}^{(r)}_{i,j,k} = \vec{W}^{(0)}_{i,j,k} - \frac{\alpha_r \Delta t}{V_{i,j,k}} \vec{R}^{(r-1)}_{i,j,k}$$

$$\vec{W}^{(n+1)}_{i,j,k} = \vec{W}^{(r)}_{i,j,k}$$

where

$$\vec{R}^{(r)}_{i,j,k} = \vec{Q}^{(r)}_{i,j,k} - \vec{D}^{(r)}_{i,j,k}, \qquad r = 0,1,2,\cdots,m = 5$$

Efficient methods to solve flow problems are well accepted and justified. Among the used approaches to obtain efficient methods, the local time-stepping, residual averaging and its combination with multigrid are employed.

Local time-stepping allow to obtain steady state solutions with less computational effort. It is equivalent to preconditioning the residual by a scalar value in each cell. It can reduce the computational time needed to obtain the steady state solution by an order of magnitude. A weighted average of residuals is employed to increase the Courant-Friedrichs-Lewy number of an explicit multistage scheme. In this way the residuals are replaced by an average of neighbouring residuals [5].

The slow asymptotic convergence behaviour of numerical methods is associated to the smooth error components. The good smoothing properties of the Runge-Kutta method, specially of the 5-stage scheme, are very important to be used in a multigrid solver. The success of the multigrid method depends on the use of a relaxation algorithm [12], which rapidly reduce the high frequency error components. The low error frequencies in fine meshes are transformed in high frequencies in coarse meshes, where they can be smoothed.

The computational procedure to illustrate the FAS (Full Approximation Storage) scheme for two grids is written as follows [12] [13]:

1. Improve the solution on the finest grid
2. Inject the flow variables from the fine to the coarse grid
3. Transfer the residuals from the fine to the coarse grid
4. Solve the problem on the coarse grid
5. Interpolate the solution correction from the coarse to the fine mesh
6. Update the solution on the finest grid

BOUNDARY CONDITIONS

The numerical treatment of boundary conditions in the physical domain is one of the major problems in solving the Euler equations. Inappropriate conditions can substantially degrade the accuracy and convergence of the computed solution. Numerical conditions imposed on the outer boundary should assure that the outgoing waves are not reflected back into the flow field.

In order to stablish an efficient numerical implementation of the boundary conditions the computational domain is surrounded by dummy cells. The body coinciding with a coordinate system is approximated by straight lines. On a solid body the physical condition of no-normal-flow can be imposed. Since the numerical treatment of the flow exterior to a body such as an airfoil requires a bounded domain, an artificial far field has to be introduced.

The approach used at far field boundaries [14] is based on the characteristic form of the one dimensional Euler equations normal to the boundary. As the far field boundary conditions assume zero circulation, they have to be placed sufficiently far away from the airfoil, so the flow field remains undisturbed.

NUMERICAL RESULTS

In the following, numerical results for a three dimensional channel and the NACA 0012 airfoil are presented and compared. A channel is chosen because it represents a class of incompressible internal flow problems. First computations were performed for channel submitted to incompressible flow. A grid containing 96x14x14 cells is used, as shown in Fig. 2. This channel, the wind tunnel of Department of Federal University of Rio Grande do Sul, is chosen in order to analyse its design. Internal honeycombs and friction are not taken into consideration in this first approximation.

Fig. 3 displays the Mach contours computed for Mach = 0.05. The corresponding vector field is presented in Fig. 4. The analysis of these results indicates good agreement with the expected solution (potential flow). It can be seen that the velocity increases with the duct reduction and the constant Mach lines remain on it.

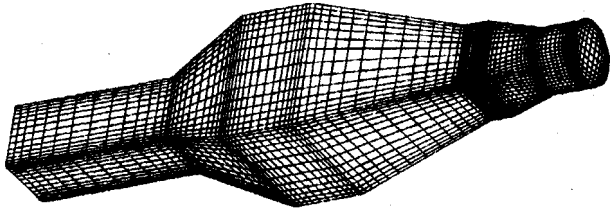

Fig. 2 Grid for channel, 96x14x14 cells (wind tunnel)

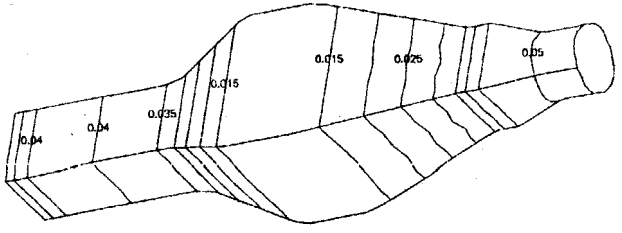

Fig. 3 Mach contours for channel, Mach = 0.05

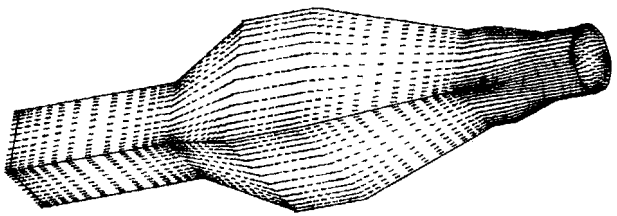

Fig. 4 Vector field for channel, Mach = 0.05

In the following, compressible and incompressible flows over the NACA 0012 airfoil are presented. Results were obtained using a C-grid topology which consists of 256x64 cells, as presented in Fig. 5. Five grids and a W-cycle are used in the multigrid process (256x64 - fine grid, 128x32 - second grid, 64x16 - third grid, 32x8 - fourth grid, 16x4 - fifth grid)

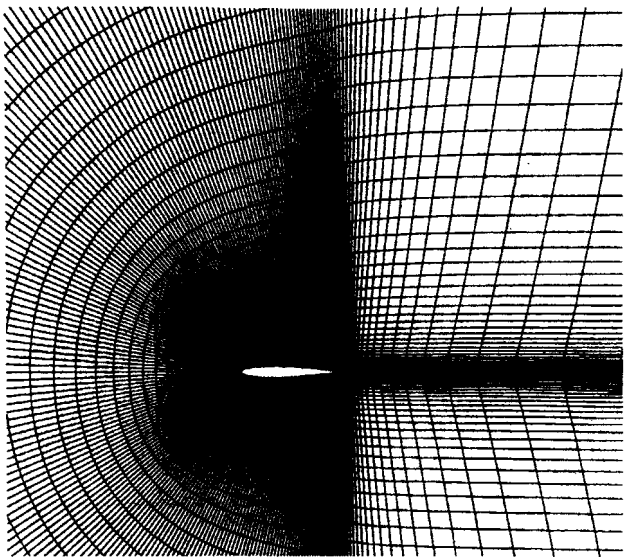

Fig. 5 Grid for NACA 0012, 256x64 cells

The position of the outer boundary is around twenty chord lengths away from the airfoil and the far field boundary condition is modified due to a vortex [15] (correction). The Euler solutions for NACA 0012 airfoil are compared to a solution obtained by the conforming mapping method [16]. Fig. 6 shows the pressure coefficient computed for Mach = 0.8 and $\alpha = 1.25°$. Fig. 7 shows the pressure coefficient computed for Mach = 0.002 and $\alpha = 5°$.

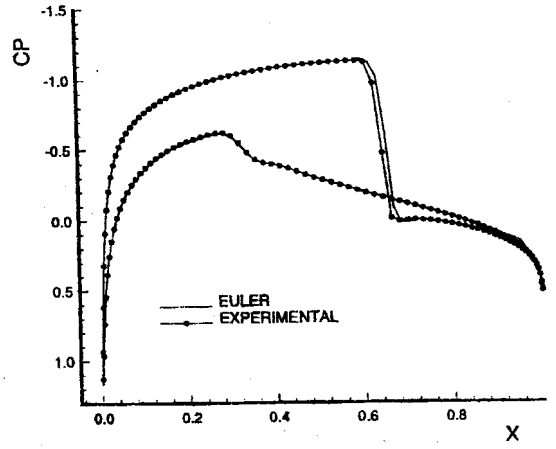

Fig. 6 Pressure coefficient for NACA 0012, Mach = 0.8 and $\alpha = 1{,}25°$

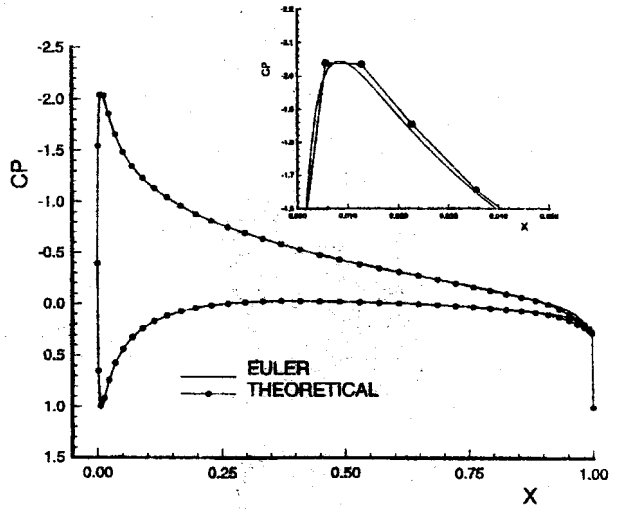

Fig. 7 Pressure coefficient for NACA 0012, Mach = 0.002 and $\alpha = 5°$

Finally, Fig. 8 compares the convergence history for Mach = 0.1 with and without the use of multigrid techniques. It turns out that the multigrid efficiency increases with the grid refinement. Besides, according to these results it is possible to solve compressible as well as incompressible flows with the present method. Pressure coefficient differences with imprecisions lower than 1% can be obtained with less than 300 time-steps.

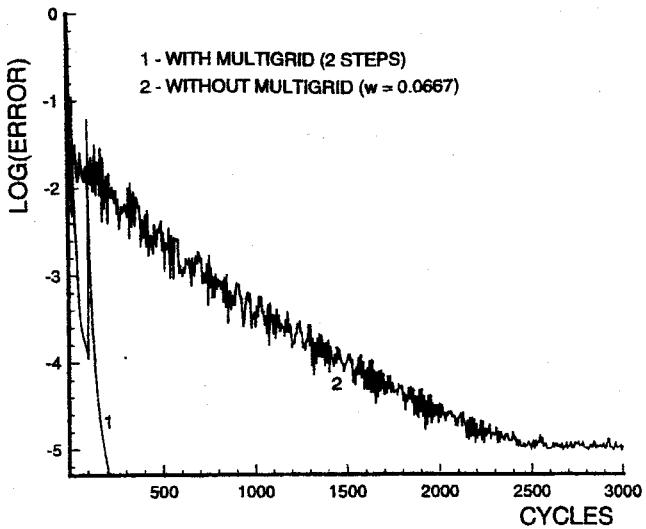

Fig. 8 Convergence history for NACA 0012 with and without multigrid techniques, Mach = 0.1

CONCLUSIONS

Numerical tests have shown that a numerical method based on the finite volume spatial discretization and the Runge-Kutta time-stepping scheme with preconditioning can be employed to solve compressible as well as incompressible fluid flows. Accuracy and reliability of the code have been tested in computing compressible and incompressible flows around airfoils. The approach is obviously vectorizable.

The preconditioning used is applicable not only to incompressible flows, but also to compressible flows with incompressible regions. Such a method allows to get the goal of compressible and incompressible solution of fluid flows. Besides, the preconditioning system has a rate of convergence almost independent of the Mach number.

Special care has been taken on the treatment of influence coefficients used to obtain the time-step, the artificial dissipation and to evaluate the far field boundary conditions. Though the far field effect of a single vortex is added to the free stream flow, is was considered necessary for subsonic flows to place the outer boundary sufficiently far away from the airfoil (20 chord lengths).

The convergence to the steady state is accelerated using local time-stepping, residual smoothing and multigrid. With these acceleration techniques the solution of the two-dimensional Euler equations can be obtained in few seconds on supercomputers for sufficient accuracy.

It is the author opinion that the comparison between the theoretical and numerical solutions is encouraging. Especially the use of multigrid techniques and its combination with preconditioning present good convergence rates. Therefore, the same code can be employed to solve compressible as well as incompressible flow problems.

ACKNOWLEDGEMENT

This work is being supported by CNPq and has been carried out at the Federal University of Rio Grande do Sul (UFRGS). Cray YMP-2E time was provided by CESUP "Centro Nacional de Supercomputação" of this university. The author is grateful to CNPq for this support and to UFRGS for this opportunity.

REFERENCES

[1] CHOI, Y.H., MERKLE, C.L., 1993, "The Application of Preconditioning in Viscous Flows", Journal of Computational Physics 105, pp 207-223.

[2] DE BORTOLI, A.L., 1994, "Solution of Incompressible Flows Using Compressible Flow Solvers", DLR IB 129-94/18, Braunschweig, Germany.

[3] VAN DOORMAAL, J.P., 1985, "Numerical Method for the Solution of Incompressible and Compressible Fluid Flows", Ph. D. Thesis, University of Waterloo, Canada.

[4] MALISKA, C.R., SILVA, A.F.C., 1989, "A Boundary Fitted Finite Volume Method for the Solution of Compressible and Incompressible Fluid Flows Using Both Velocity and Density Corrections", Seventh International Conference on Finite Element Methods in Flow Problems, Hunstwille.

[5] JAMESON, A., SCHMIDT, W., TURKEL, E., 1981, "Numerical Solution of the Euler Equations by Finite Volume Methods Using Runge-Kutta Time-Stepping Schemes", AIAA Paper 81-1259.

[6] KROLL, N., JAIN, R.K., 1987, "Solution of Two-Dimensional Euler Equations-Experience with a Finite Volume Code", Forschungsbericht, DFVLR-FB 87-41, Braunschweig, Germany.

[7] CHOI, D., MERKLE, CL., 1985, "Application of Time-Iterative Schemes to Incompressible Flow", AIAA Journal, Vol. 23, Nr. 10, pp. 1518-1524.

[8] STORTI, M., NIGRO, N., IDELSOHN, S., 1995, "Steady State Incompressible Flows Using Explicit Schemes with an Optimal Local Preconditioning", Comp. Methods Appl. Mech. Engrg. 124, pp. 231-252.

[9] KROLL, N., ROSSOW, C.-C., 1989, "Foundations of Numerical Methods for the Solution of Euler Equations", Prepared for the Lecture F 6.03 of the CCG, Braunschweig, Germany.

[10]BLAZEK, J., 1994, "Verfahren zur Beschleunigung der Lösung der Euler- und Navier-Stokes Gleichungen bei Stationären Über- und Hyperschallströmungen", Ph.D. Thesis, University of Braunschweig.

[11]RADESPIEL, R., 1989, "A Cell-Vertex Multigrid Method for the Navier-Stokes Equations", NASA Technical Memorandum 101557.

[12]BRANDT, A., 1981, "Guide to Multigrid Development", Multigrid Methods I, Lecture Notes in Mathematics.

[13]JAMESON, A., 1985, "Multigrid Algorithm for Compressible Flow Calculations", Multigrid Methods, Lecture Notes in Mathematics, Cologne, Germany.

[14]WHITFIELD, D.L., 1983, "Three-Dimensional Unsteady Euler Equation Solutions Using Flux Vector Splitting", MS 39762, Mississippi State University.

[15]USAB, W.J., MURMAN, E.M., WHITFIELD, D.L., 1983, "Embedded Mesh Solution of the Euler Equation Using a Multiple-Grid Method", AIAA Paper 83-1946-CP.

[16]BLAZEK, J., 1989, "Berechnung der Strömung um Profilsysteme mit und ohne Bodeneffect mit der Methode der Konformen Abbildung", M. Sc. Dissertation, RWTH Aachen.