Mecánica Computacional Volumen XV
Compilado por A. Larreteguy y M. Vénere
Bariloche, noviembre de 1995

A NUMERICAL METHOD FOR THE SOLUTION OF COMPRESSIBLE AND INCOMPRESSIBLE FLUID FLOWS

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

MALISKA, C.R.:
Department of Mechanical Engineering
Federal University of Santa Catarina
P.O. Box 476 - 88040-900 - Florianópolis - SC- Brazil

ABSTRACT

Numerical simulation plays nowadays an important role to predict the flow field in many situations. To design a new mechanical device involving fluid dynamics, a numerical simulation is well accepted and justified. However, many work still remains to improve the numerical methods towards a fast, accurate and stable convergence. This work presents efficiency studies to solve compressible and incompressible fluid flows using a finite-volume, explicit Runge-Kutta multistage scheme, with central spatial discretization in combination with multigrid. An extension of the methodology normally employed to solve compressible flows is used to solve incompressible flow problems. Numerical results are presented for a cylinder and the NACA 0012 airfoil for Mach-numbers ranging from 0.8 to 0.005 using the Euler equations.

INTRODUCTION

Today, numerical flow simulation plays more and more important role in the design process of an aerodynamic body. It is already possible to solve the flow over a complete aircraft geometry for specific flow regimes. The interest now is to develop efficient methodologies that can be used to solve all speed flow problems. As the use of such methodologies is limited, these limitations must be investigated in order to design at low costs.

Physical and mathematical principles are important tools to analyse these problems. As the analytical methods are limited, numerical methods allow us to analyse some important phenomena, where experiments are usually too expensive or even impossible. In this way one strategy is to approximate the governing equations of the problem.

There are well known methods to solve either compressible and incompressible flow problems. When solving high speed flows it is common to approximate the differential equations using high order interpolation functions [1] [2]. In order to turn the numerical procedure more stable artificial dissipation is needed. The solution of low speed flows, however, is usually obtained using hybrid interpolation functions, that are based on the physical aspects of flow behaviour [3] [4].

Extensions of the methodologies employed for incompressible flows have been applied, with success, for the solution of all speed flow problems [5] [6] [7]. One drawback of these methods was the use of staggered variables in order to provide an adequate pressure-velocity/density coupling. This problem was partially solved using the colocated arrangement

[8] [9] [10]. Other drawback of these methods is their accuracy and cost when solving transonic flow problems over aerodynamic geometries.

Extensions of the methodologies used to solve compressible flows have also been applied to solve all speed flows [11] [12]. Sometimes these methods are based on the introduction of artificial density and are denoted as β-compressibility methods [13].

The potential equations were and sometimes are still used to solve compressible and incompressible flow problems. This is done in order to simplify the analysis and to reduce the computational costs. However, there are problems that can not be analysed using the potential flow equations, for example flows with strong shocks and with vorticity. The only adequate model for non viscous flows are the Euler equations, in which mass, momentum and energy are conserved.

In order to efficiently solve problems with fine grids, techniques to accelerate the convergence to the steady state are required. Some of these approaches, namely the local time-stepping, residual averaging and multigrid techniques [14] [15] [16] are normally employed. The first allows to obtain steady state solutions with less computational effort. Residual averaging is used to increase the Courant number of an explicit scheme and consists in replacing the residuals by an average of neighbouring residuals. The idea of the multigrid approach is to use a sequence of successively coarser meshes to efficiently damp disturbances in the flow field.

This work presents studies to solve compressible and incompressible fluid flows using the finite volume explicit Runge-Kutta multistage scheme with central spatial discretization in combination with multigrid. It is an extension of the work developed in [17] specially for high speed flow problems. Numerical results are presented for cylinder and the NACA 0012 airfoil for Mach-numbers ranging from 0.8 to 0.005 [18] using the Euler equations. This is done in order to show that the same methodology can be employed to solve compressible as well as incompressible flow problems.

GOVERNING AND APPROXIMATE EQUATIONS

The governing equations for non viscous flows are the Euler equations. They can be written in cartesian coordinates for bidimensional problems as:

$$\frac{\partial \vec{W}}{\partial t}+\frac{\partial \vec{F}}{\partial x}+\frac{\partial \vec{G}}{\partial y}=0 \tag{1}$$

where

$$\vec{W}=\begin{Bmatrix}\rho\\ \rho u\\ \rho v\\ \rho E\end{Bmatrix},\qquad \vec{F}=\begin{Bmatrix}\rho u\\ \rho uu+p\\ \rho uv\\ \rho H u\end{Bmatrix},\qquad \vec{G}=\begin{Bmatrix}\rho v\\ \rho uv\\ \rho vv+p\\ \rho H v\end{Bmatrix}$$

The total energy and total enthalpy are

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

To close this system of equations the state equation for a perfect gas is used

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

where R is the universal gas constant, ρ the density, u and v the velocity components and p the pressure. The governing equations can also be cast into the integral form [2]

$$\int_V \frac{\partial \vec{W}}{\partial t} dV + \int_S (\overline{F}.\vec{n}) dS = 0 \qquad (3)$$

As the governing equations are valid for an arbitrary control volume, they are also valid for the quadrilateral $\bar{V}_{i,j}$, and can be approximated as follows

$$\frac{\partial \vec{W}_{i,j}}{\partial t} = -\int_S (\overline{F}.\vec{n}) dS = -\frac{1}{V_{i,j}} \vec{Q}_{i,j} \qquad (4)$$

This form of conservation laws allows discontinuities and is consistent. However, 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. Then preconditioning is employed in order to assure rapid convergence. It consists basically in multiplying the vector $\vec{W}$ by a special matrix, that modifies the general form of the governing equations [11]. Based on the conservative variables, the following preconditioning matrix is employed

$$\Gamma \frac{\partial \vec{W}}{\partial t} + \frac{\partial \vec{F}}{\partial x} + \frac{\partial \vec{G}}{\partial y} = 0 \qquad (5)$$

where [11]

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

Inspection of this preconditioning matrix indicates that the energy equation is transformed into an equation for 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 [12] to solve incompressible flows [18].

TIME-STEPPING SCHEME

The system of governing equations is discretized separately in time and space. The discretization follows the finite volume method. A modified Runge-Kutta time-stepping scheme [1] [2] is chosen, because its stage coefficients can be tuned in order to optimise the damping of transient disturbances, which is important for application of a multigrid method. As the classical fourth order Runge-Kutta method requires the evaluation of convective fluxes $\vec{Q}_{i,j}^{(k)}$ and dissipative fluxes $\vec{D}_{i,j}^{(k)}$ for each stage (m-stage; $k = 0, 1, ..., m$), this scheme leads to storage problems. A simplified Runge-Kutta scheme, requiring lower storage is given by [2]

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

where

$$\vec{R}_{i,j}^{(k)} = \vec{Q}_{i,j}^{(k)} - \vec{D}_{i,j}^{(k)}$$

The dissipation terms $\vec{D}_{i,j}$ are introduced by adding dissipative fluxes, which preserve the conservative form, to the semi-discrete system written as [1]

$$\frac{d\vec{W}_{i,j}}{dt} + \frac{1}{V_{i,j}}\left[\vec{Q}_{i,j} - \vec{D}_{i,j}\right] = 0 \qquad (6)$$

For steady state problems the stage coefficients can be selected purely with respect to stability and damping properties. The following coefficients lead to an efficient 5-stage scheme

$$\alpha_1 = 1/4, \quad \alpha_2 = 1/6, \quad \alpha_3 = 3/8, \quad \alpha_4 = 1/2, \quad \alpha_5 = 1$$

NUMERICAL RESULTS

In the following, numerical results for a cylinder and the NACA 0012 are presented and compared. One way of proving the validity of the numerical Euler solutions is to compare them with potential solutions.

First computations were performed for incompressible flow over a cylinder. A O-grid topology with 160x48 cells is used. The position of the outer boundary is around 40 diameters away from the cylinder.

Fig. 2. shows the pressure contours computed for Mach = 0.05. The corresponding pressure coefficient is presented in Fig. 3. The analysis of these results indicates good agreement with the analytical solutions [17].

In the following, compressible and incompressible flows over the NACA 0012 airfoil are presented. Results were obtained using a C-grid topology that consists of 256x64 cells. The position of the outer boundary is around twenty chord length away from the airfoil and the far field boundary condition is modified due to a vortex [2] [17] [18]. Five grids were employed in the multigrid process.

First computations were performed at Mach = 0.8. Fig. 5 shows the pressure contours computed for Mach = 0.8 and α = 0°. Fig. 6 shows the pressure coefficient computed for Mach = 0.005. The analysis of these results indicates that the present methodology can be used to solve compressible as well as incompressible flows accurately. Fig. 7 indicates that two time steps and a W-multigrid cycle (with 4 coarse grids) are advised in the multigrid process for low Mach-numbers. Fig. 8 compares the convergence history for Mach = 0.1 with and without the use of multigrid techniques. The computational time can be reduced by a factor of 10 when using this technique.

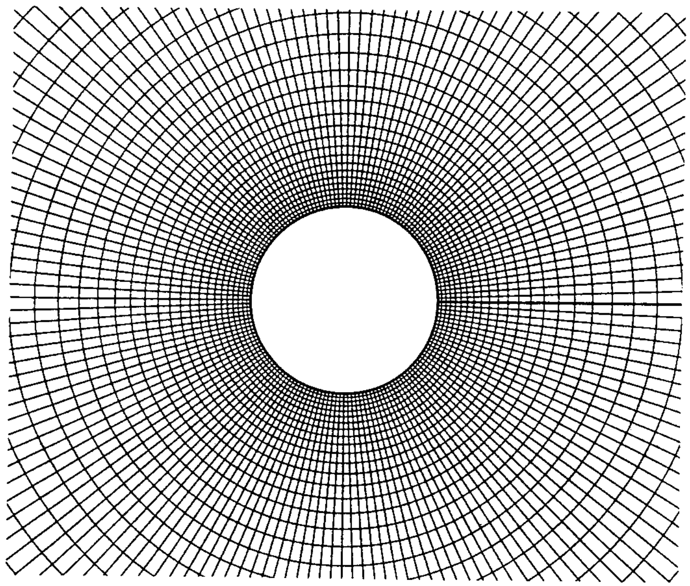

Fig. 1 Grid for cylinder, 160x48 cells

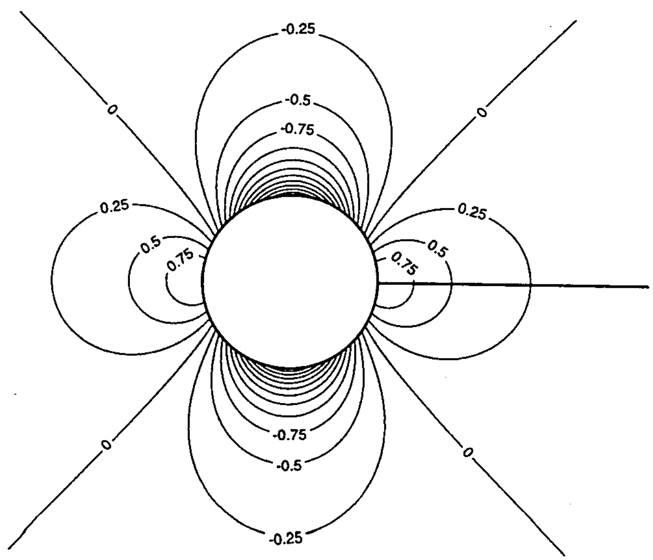

Fig. 2 Pressure contours for cylinder, Mach = 0.05

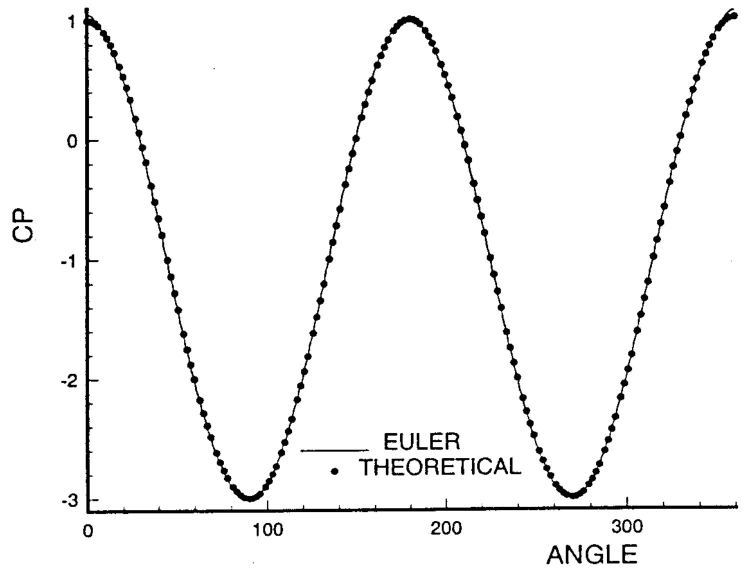

Fig. 3 Pressure coefficient for cylinder, Mach = 0.05

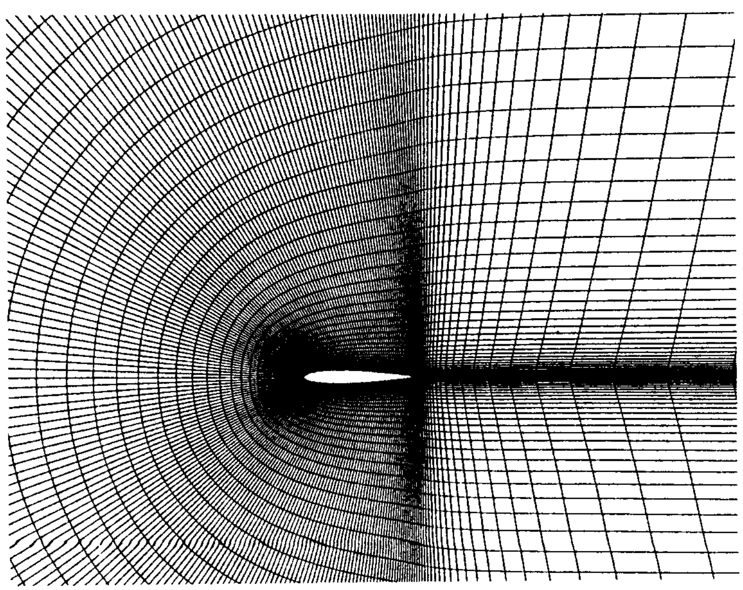

Fig. 4 Grid for NACA 0012, 256x64 cells

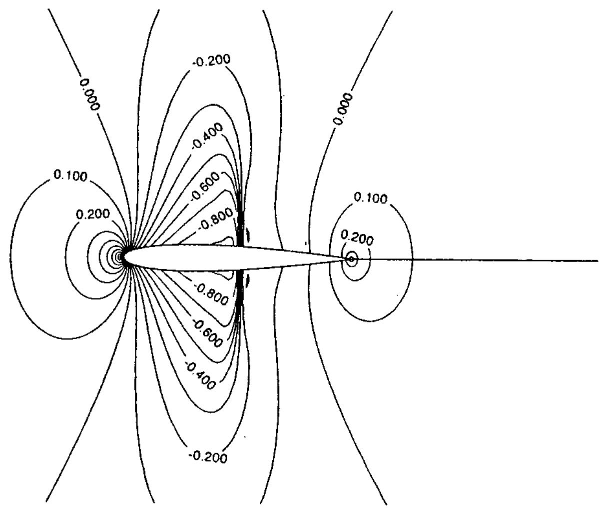

Fig. 5 Pressure contours for NACA 0012, Mach = 0.8 and $\alpha = 0°$

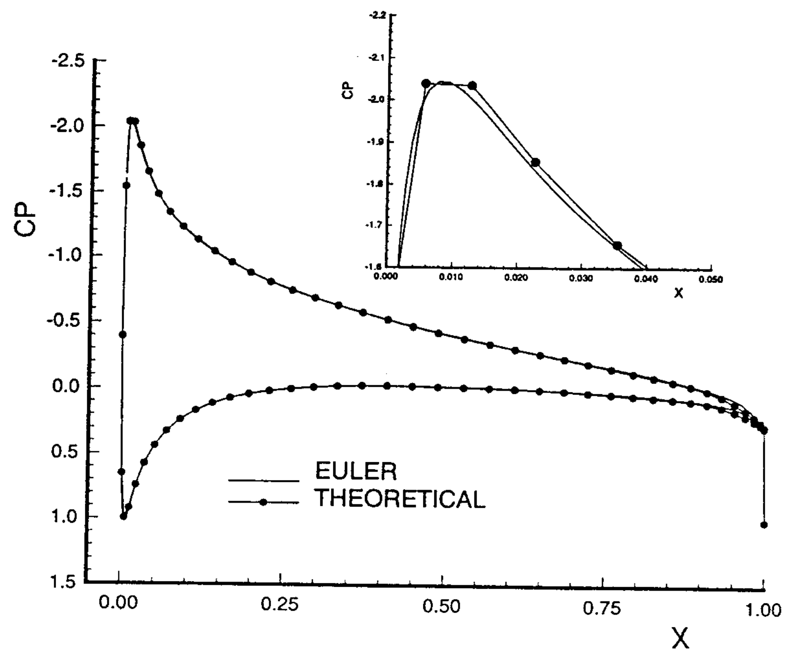

Fig. 6 Pressure coefficient for NACA 0012, Mach = 0.005 and $\alpha = 5°$

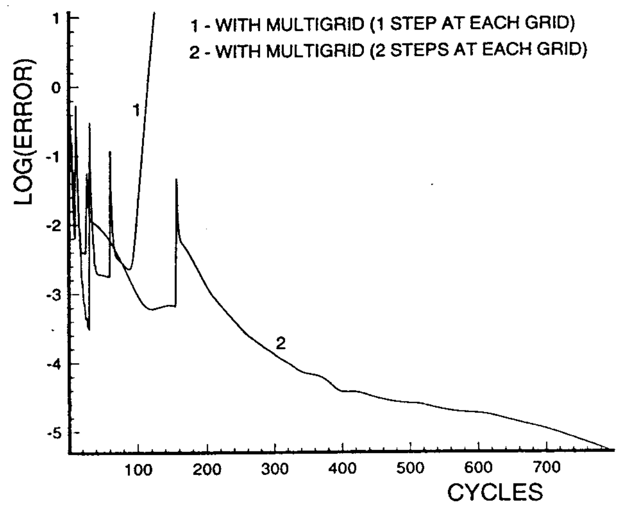

Fig. 7 Convergence histories for NACA 0012 for Mach 0.01 and $\alpha = 5°$

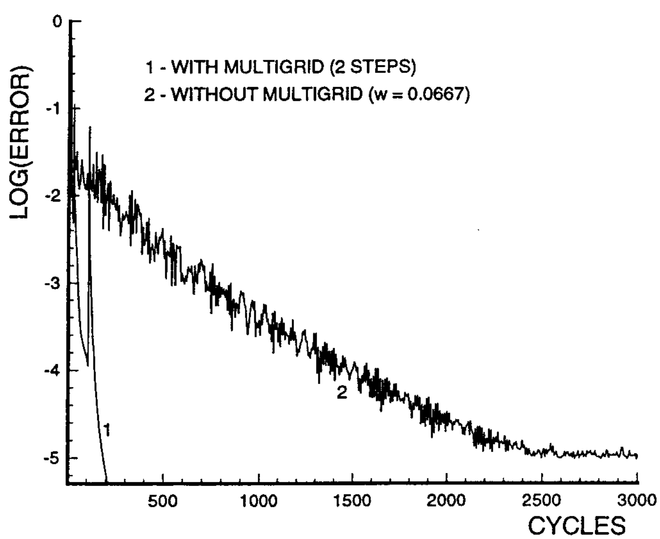

Fig. 8 Convergence history for NACA 0012 with and without multigrid techniques, Mach = 0.1

CONCLUSIONS

Extension of a compressible code [17], based on the node-centered arrangement, to solve incompressible flows is presented and compared for a cylinder and the NACA 0012 airfoil. The finite volume spatial discretization and the Runge-Kutta time-stepping scheme are used to efficiently solve compressible as well as incompressible flow problems. Special care has been taken on the treatment of influence coefficients used to obtain the time-step and the artificial dissipation.

Present numerical results permit to conclude that it is possible and preferable to accelerate the convergence to obtain steady state solutions using the multigrid technique. However, for low Mach flows the efficiency of this technique was not so big as for transonic flows presented by [17]. Two time-steps were necessary at each grid and the W-multigrid cycle in order to efficiently use the multigrid approach for the solution of small speed flows.

The comparison between the theoretical and numerical solutions is encouraging. Especially the use of multigrid techniques and its combination with preconditioning leads to reasonable rates of convergence. Besides, the same code can be employed to solve compressible as well as almost incompressible flow problems.

ACKNOWLEDGEMENT

This work was supported by DAAD (Deutscher Akademischer Austauschdienst) and carried out at the DLR Institute of Design Aerodynamics in Braunschweig. The author is grateful to DAAD for this support and to DLR for this opportunity. The author gratefully acknowledges numerous discussions with Dr. N. Kroll and Dr. J. Blazek during this work.

REFERENCES

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

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

[3] PATANKAR, S.V., Numerical Heat Transfer and Fluid Flow, McGraw-Hill, New York, 1981.

[4] RAITHBY, G.D., SCHNEIDER, G.E., Numerical Solution of Problems in Incompressible Flow: Treatment of Velocity-Pressure Coupling, Numerical Heat Transfer, Vol. 2, 1979, pp 417-440.

[5] HARLOW, F.H., AMSDEM, A.A., A Numerical Fluid Dynamics Calculation for All Flow Speeds, J. Comp. Physics, Vol. 8, 1971, pp 197-213.

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

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

[8] RHIE, C.M., A Numerical Study of the Flow Past an Isolated Airfoil with Separation, Ph. D. Thesis, University of Illinois at Urbana-Champaign, 1981.

[9] PERIC, M., KESSLER, R., SCHEURER, G., Comparison of Finite-Volume Numerical Methods with Staggered and Colocated Grids, Computer and Fluids, Vol. 16, 1988, pp. 389-403.

[10] MARCHI, C.H., MALISKA, C.R., DE BORTOLI, A.L., The Use of Colocated Variables in the Solution of Supersonic Flows, X Brazilian Congress of Mechanical Engineering, December, 1989.

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

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

[13] TURKEL, E., Review of Preconditioning Methods for Fluid Dynamics, NASA Report 189712, ICASE Report Nr. 92-47, 1992.

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

[15] JAMESON, A., Multigrid Algorithm for Compressible Flow Calculations, Multigrid Methods, Cologne, October 1985.

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

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

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