A NUMERICAL METHOD FOR COMPUTING FLOWS FOR ARBITRARILY SMALL AND MEDIUM MACH NUMBERS

H. BIJL AND P. WESSELING
*Delft University of Technology,
Faculty of Technical Mathematics and Informatics,
PO Box 5031, 2600 GA Delft, The Netherlands*

1. Introduction

We have developed a method for the numerical computation of flows with arbitrarily small and medium Mach numbers. This method is accurate and efficient for Mach numbers ranging between 0 and $\mathcal{O}(1)$. The equations are nondimensionalised such that all quantities remain finite for all Mach numbers. Discretisation is done in general coordinates on a staggered grid, using an upwind-biased density evaluation when the local Mach number comes above 0.9. The system is solved iteratively with the pressure correction method. The discretisation and solution method become identical to the classical incompressible method [4] as the Mach number tends to zero. Mach number independent accuracy and convergence is confirmed by numerical experiments.

2. Governing Equations

Adiabatic compressible flow of inviscid fluids is considered, so that the flow is governed by the Euler equations. We use the pressure as primary unknown instead of the density since in the incompressible limit the density becomes constant, whereas the pressure can still vary. In Cartesian tensor notation the equations are given by

$$\text{continuity eq.} \quad \left(\frac{\partial \rho}{\partial p}\right)_h \frac{\partial p}{\partial t} + \left(\frac{\partial \rho}{\partial h}\right)_p \frac{\partial h}{\partial t} + (\rho u^\alpha)_{,\alpha} = 0 \quad (1)$$

$$\text{momentum eq.} \quad \frac{\partial \rho u^\alpha}{\partial t} + (\rho u^\alpha u^\beta)_{,\beta} = -p_{,\alpha} \quad (2)$$

$$\text{energy eq.} \quad \frac{\partial h}{\partial t} + (h u^\alpha)_{,\alpha} = -(\gamma - 2) h u_{,\alpha}^\alpha \quad (3)$$

where $u^\alpha = u_\alpha$ are the Cartesian velocity components, ρ is the density, p is the pressure, and h is the enthalpy. The equation of state completes the system of equations:

$$\rho = \frac{\gamma}{\gamma - 1} \frac{p}{h}. \quad (4)$$

We will discuss internal flow in a two-dimensional channel or nozzle, the inflow and outflow boundaries are referred to by subscripts ∞ and out , respectively. When the inflow is subsonic, the boundary conditions are:

$$\begin{aligned} u_\infty^\alpha &= u_\infty^\alpha(y, t) = \text{given}, \quad \alpha = 1, 2 \\ h_\infty &= h_\infty(y, t) = \text{given}, \\ p_{out} &= p_{out}(y, t) = \text{given}. \end{aligned}$$

For supersonic inflow, all variables should be specified on the inlet boundary, so instead of the pressure at the outflow, $p_\infty = p_\infty(y, t) = \text{given}$. At a supersonic outlet boundary all variables are linearly extrapolated from the inner region. On solid boundaries the impermeability condition is prescribed: $u_\alpha n^\alpha = 0$ with $\mathbf{n}$ the outward normal on the solid wall. The initial conditions specify p , u^α , and h .

3. Nondimensionalisation

The variables are non-dimensionalised such that they remain non-zero and finite in the limit of the characteristic Mach number $M_{ref} \rightarrow 0$. In this limit we want the non-dimensionalised Euler equations to be well defined and to reduce to the incompressible Euler equations too. This is not the case for standard compressible flow computing methods, which as a consequence break down for low Mach numbers. It is assumed that the scale of the boundary conditions is characteristic for the magnitude of the variables. Therefore the Mach number at the inlet M_∞ is chosen as reference Mach number M_{ref} . For a motivation of the units adopted see [2]. The nondimensional variables are:

$$\begin{aligned} x^1 &= \frac{x^{1*}}{L^*} & x^2 &= \frac{x^{2*}}{L^*} \\ t &= \frac{t^* w_\infty^*}{L^*} & p &= \frac{p^* - p_{out}^*}{\rho_0^* w_\infty^{*2}} \\ h &= \frac{h^*}{h_0^*} & \rho &= \frac{\rho^*}{\rho_0^*} \\ u^1 &= \frac{u^{1*}}{w_\infty^*} & u^2 &= \frac{u^{2*}}{w_\infty^*}, \end{aligned}$$

where $*$ indicates that variable is dimensional. Furthermore, w_∞^* is the speed at the inlet, L^* a reference length and p_{out} the pressure at the outlet. As

reference enthalpy and density the stagnation conditions are chosen:

$$h_0^* = \left(1 + \frac{\gamma-1}{2} M_\infty^2\right) h_\infty^*, \quad (5)$$

$$\rho_0^* = \left(1 + \frac{\gamma-1}{2} M_\infty^2\right)^{\frac{1}{\gamma-1}} \rho_\infty^*. \quad (6)$$

The nondimensional Euler equations are in this case the same (1)-(3). The nondimensional equation of state becomes:

$$\begin{aligned} \rho = \rho(p, h) &= \frac{\gamma}{\gamma-1} \frac{w_\infty^{*2}}{h_0^*} \frac{p}{h} + \frac{\gamma}{\gamma-1} \frac{p_{out}^*}{\rho_0^* h_0^*} \frac{1}{h} = \\ &= \frac{\gamma M_\infty^2}{1 + \frac{\gamma-1}{2} M_\infty^2} \frac{p}{h} + \left[p_v \left(\left(1 + \frac{\gamma-1}{2} M_\infty^2\right)^{\frac{-\gamma}{\gamma-1}} - 1 \right) + 1 \right] \frac{1}{h}, \end{aligned} \quad (7)$$

where p_v is defined by:

$$p_v = \frac{p_{out}^* - p_0^*}{p_\infty^* - p_0^*}. \quad (8)$$

The non-dimensional subsonic boundary conditions are:

$$\begin{aligned} (\rho u)_\infty^1 &= \left(1 + \frac{\gamma-1}{2} M_\infty^2\right)^{\frac{-1}{\gamma-1}} \cos \alpha_\infty, \\ (\rho u)_\infty^2 &= \left(1 + \frac{\gamma-1}{2} M_\infty^2\right)^{\frac{-1}{\gamma-1}} \sin \alpha_\infty, \\ h_\infty &= \left(1 + \frac{\gamma-1}{2} M_\infty^2\right)^{-1}, \\ p_{out} &= 0, \end{aligned} \quad (9)$$

where α_∞ is the inlet flow angle. For supersonic inflow p_∞ must be prescribed instead of p_{out} . For the boundary condition on the momentum components M_∞ and α_∞ need to be known, while the other two boundary conditions follow directly from the scaling used. The nondimensional problem can now be solved when M_∞ , α_∞ , and p_v are given; M_∞ and α_∞ are necessary for the nondimensional boundary conditions, and p_v appears in the scaled equation of state. Note that the solution is independent of ρ_∞^* , but to recover the original variables from the nondimensional ones we need both h_∞^* and ρ_∞^* .

4. Invariant formulation

For computing flows in general domains the governing equations are recast in a coordinate-invariant form. The compressible Euler equations in Cartesian coordinates of (1), (2), and (3) are not tensor equations because α

occurs both as a free superscript and a free subscript, and we have a pair of equal subscripts. Raising subscripts as required by contraction with the metric tensor [1] we obtain tensor equations, which are then valid in general coordinates, according to Ricci's lemma:

$$\left(\frac{\partial \rho}{\partial p}\right)_h \frac{\partial p}{\partial t} + \left(\frac{\partial \rho}{\partial h}\right)_p \frac{\partial h}{\partial t} + (\rho U)^\alpha_{,\alpha} = 0 \quad (10)$$

$$\frac{\partial(\rho U)^\alpha}{\partial t} + ((\rho U)^\alpha U^\beta)_{,\beta} = -(g^{\alpha\beta} p)_{,\beta} \quad (11)$$

$$\frac{\partial h}{\partial t} + U^\alpha h_{,\alpha} = -(\gamma - 1)h U^\alpha_{,\alpha} \quad (12)$$

with $(\rho U)^\alpha = \mathbf{a}^{(\alpha)} \cdot (\rho \mathbf{u})$ the contravariant momentum components, where the contravariant metric tensors are defined as

$$g^{\alpha\beta} = \mathbf{a}^{(\alpha)} \cdot \mathbf{a}^{(\beta)}, \quad \mathbf{a}^{(\alpha)} = \frac{\partial \xi^\alpha}{\partial \mathbf{x}} \quad (13)$$

the contravariant base vectors with respect to the mapping $T : \mathbf{x} = \mathbf{x}(\xi)$. Here, $\mathbf{x}$ are the Cartesian coordinates in the physical domain Ω and ξ are the boundary-conforming curvilinear coordinates in the computational domain G . The mapping is assumed to be regular, i.e. the Jacobian of the transformation, that will be denoted as $\sqrt{g}$, does not vanish. For more details see [6, 9].

Figure 1. Cell in computational domain G and its image in the physical domain Ω

5. Discretisation in space

The compressible Euler equations will be discretised in space in boundary-fitted coordinates using a finite volume technique on a staggered grid. We have chosen for a staggered grid because of the following advantages for incompressible flow. To begin with, the differential equations can be discretised in a natural and straightforward way. Furthermore, the physical boundary conditions suffice and no measures have to be taken to avoid spurious pressure oscillations. The main difference with methods for the incompressible case is that the density is an unknown variable, which results in an extra term in the continuity equation and an extra equation:

the equation of state. Our method will be chosen such that as $M_\infty \downarrow 0$ the classical incompressible staggered grid method [4] is recovered. This may be expected to give Mach-independent convergence for small and medium Mach numbers. Figure 1 shows part of the computational grid with the staggered placement of the unknowns and the corresponding control volumes. The discretisation in general coordinates introduces mass flux components ρV^α , $V^\alpha = \sqrt{g} U^\alpha$. For more details on invariant discretisation in general coordinates, see [5].

6. Pressure Correction Method

Fully implicit discretisation in time of the continuity and momentum equations gives:

$$\left(\frac{\partial \rho}{\partial p}\right)_h^{n+1} \frac{p^{n+1} - p^n}{\delta t} + \left(\frac{\partial \rho}{\partial h}\right)_p^{n+1} \frac{h^{n+1} - h^n}{\delta t} + (\rho U^\alpha)_{,\alpha}^{n+1} = 0 \quad (14)$$

$$\frac{(\rho U^\alpha)^{n+1} - (\rho U^\alpha)^n}{\delta t} + \left((\rho U^\alpha)^{n+1} (U^\beta)^{n+1}\right)_{,\beta} = - \left(g^{\alpha\beta} p^{n+1}\right)_{,\beta}. \quad (15)$$

An efficient way to solve this system is the pressure correction method a standard workhorse for incompressible flows [4]. In this method, first, a predictor momentum field $(\rho U^\alpha)^*$ is computed from

$$\frac{(\rho U^\alpha)^* - (\rho U^\alpha)^n}{\delta t} + \left((\rho U^\alpha)^* (U^\beta)^n\right)_{,\beta} = - \left(g^{\alpha\beta} p^n\right)_{,\beta}, \quad (16)$$

where the Picard method is used for the linearisation of the convection term. Next a pressure correction $\delta p = p^{n+1} - p^n$ is computed. To find this correction equation, first (16) is subtracted from (15) neglecting the difference in the convection term:

$$\frac{(\rho U^\alpha)^{n+1} - (\rho U^\alpha)^*}{\delta t} = - \left(g^{\alpha\beta} (p^{n+1} - p^n)\right)_{,\beta}, \quad (17)$$

Van Kan [7] has shown for the incompressible case that the effect of this neglect on the accuracy is for the Crank-Nicholson method negligible, the resulting velocity at the new time level remains $\mathcal{O}(\delta t^2)$ accurate. Then the divergence of (17) is taken; with substitution of (14), taking the coefficients $\left(\frac{\partial \rho}{\partial p}\right)$ and $\left(\frac{\partial \rho}{\partial h}\right)$ at the previous time level, this gives:

$$\left(\frac{\partial \rho}{\partial p}\right)_h^n \frac{\delta p}{\delta t} + \left(\left(g^{\alpha\beta} \delta p\right)_{,\beta}\right)_{,\alpha} = \frac{(\rho U^\alpha)^*_{,\alpha}}{\delta t} - \left(\frac{\partial \rho}{\partial h}\right)_p^n \frac{\delta h}{\delta t} \quad (18)$$

When $\delta h = h^{n+1} - h^n$ is known, the pressure correction δp can be computed from (18), whereafter $(\rho U^\alpha)^{n+1}$ can be found from (17). This δh can be

computed from the following discretisation of the energy equation, which is implicit in h but explicit in the other unknowns:

$$\frac{h^{n+1} - h^n}{\delta t} + (U^\alpha)^n h_{,\alpha}^{n+1} = -(\gamma - 1)h^{n+1} (U^\alpha)_\alpha^n. \quad (19)$$

To obtain at the end of a time step the velocity components $(U^\alpha)^{n+1}$ from $(\rho U^\alpha)^{n+1}$, these mass fluxes have to be divided by the density ρ^{n+1} , which is computed from the equation of state $\rho^{n+1} = \rho(p^{n+1}, h^{n+1})$ see (7). In practice the Euler equations are first discretised in space before the pressure correction method is applied, so that the equations derived in this section are linear systems of discrete equations. These systems were solved by a Krylov subspace iterative method for unsymmetric matrices, namely GMRES, see [8].

7. Numerical results

First flow in a channel with a 10% bump is computed. The size of the channel is $[0, 3] \times [0, 1]$, in which a boundary fitted nonuniform grid of 63×22 cells is generated. We use time-stepping to obtain a steady solution. For this the following termination criterion is used: $\max_{i,j} ((u_{i,j}^\alpha)^{n+1} - (u_{i,j}^\alpha)^n) \leq \delta t 10^{-6}$. Since, the velocity components, $u_{i,j}^\alpha$, are non-dimensionalised by the inlet speed, scaling of the termination criterion is unnecessary. As initial conditions for the time iteration, we use $\rho U^1 = \rho U^2 = 0, h = h_\infty, p_s = 0$. Computations were carried out for the following inlet Mach numbers: $M_\infty = 10^{-6}, 0.01, 0.1, 0.5$, and 0.675 . The boundary conditions, described in Chapter 3, depend on three parameters: the inlet Mach number, the angle of the flow at the inlet, and the pressure ratio p_v . For the channel with bump the last two parameters were chosen to be: $\alpha_\infty = 0, p_v = 1$. For an inlet Mach number of $M_\infty = 0.5$ the resulting iso-Mach lines are shown in Figure 2. Note that the picture is symmetric, as it should be for subsonic flow through the specified channel. The computed Mach number distributions on the upper and lower boundary of the channel are compared to results obtained in [3] in Figure 2. In the channel with a 10% bump transonic flow arises at an inlet Mach number of 0.675 . The location of the shock was at 70% of the bump, where [3] found 72 %. The maximum Mach number we found was 1.33 against 1.32 in [3]. At the bump, the shock is captured in 4 cells.

The second test application was a converging/diverging nozzle with the following contraction ratios: 5, 10, and 20. The inlet Mach number is kept constant at 0.045. The size of the throat is kept constant at 0.4, and the size of the outlet is kept constant at 2.5 times the size of the throat. For all computations we have used a 49×10 nonuniform grid. The time step

was kept constant at 0.01. In the computations for flow in nozzles with contraction ratios 5 and 10, $p_v = 1$ was used, and the flow remained subsonic. The maximum Mach numbers that occurred are listed in Table 2. For a contraction ratio of 20, $p_v = 60$ was chosen so that supersonic outflow occurred. In this case upwind biased density evaluation was used to ensure thermodynamic irreversibility to satisfy the entropy condition.

Figure 2. Mesh and Mach number plots for $M_\infty = 0.5$, and 0.675.

The number of iterations and CPU time that were necessary in both cases are listed in Table 1. For the channel with bump neither the number of iterations nor the CPU time deteriorate for low inlet Mach numbers: also at low Mach numbers a good solution is obtained efficiently. Only for increasing Mach numbers the CPU time increases a little; this might be due to the fact that local Mach numbers become as high as .728 while we have not used any density upwinding here. In the nozzle with a contraction ratio of 5 the flow remained incompressible. The maximum Mach number that occurred was 0.24, so almost equal to 5 times the inlet Mach number. For a contraction ratio of 10 the maximum Mach number was 0.5, so the Mach number varied in this flow from 0.045 up to 0.5. In the case of a contraction ratio of 15 this range was from 0.045 up to 1.82. The CPU time and number of iterations for the nozzle do not increase significantly for lower Mach numbers, that is smaller contraction ratios, either. On the contrary, there is a larger increase of computing time for the contraction ratio 20 than for the contraction ratio 5 compared to 10. This increase of CPU time is caused by an increase of the number of time steps necessary to reach the stationary solution, which is probably due to the fact that we have kept the time step constant.

8. Concluding Remarks

A method to compute inviscid flow has been described, that has accuracy and efficiency uniform for very low and medium Mach numbers. A non-

TABLE 1. Channel with 10% bump.

Mach nr.	CPU [s]	time steps	GMRES iter/t.st.
0.5	151	509	12
0.1	73	220	18
0.01	74	200	21
10^{-6}	74	200	21

TABLE 2. Converging/diverging nozzle.

contr. ratio	max. Mach	CPU [s]	time steps	GMRES iter/t.st.
15	1.82	750	6790	14
10	.5	220	1866	14
5	.24	300	2744	14

dimensionalisation based on the reservoir parameters has been introduced such that all quantities in the scaled compressible Euler equations remain finite and $\mathcal{O}(1)$ when the Mach number goes to zero. The nondimensional equations are discretised in general coordinates on a staggered grid and solved with a pressure correction method and implicit time stepping. From the numerical experiments the following conclusions can be drawn:

- Good results are obtained for subsonic flow at low Mach numbers, even solutions at $M = 0$ are no problem.
- Mach number independent convergence is observed for subsonic flow.
- Our method is able to compute fully compressible transonic flow.
- Locations of shocks are predicted well.

References

1. R. Aris, *Vectors, tensors and the basic equations of fluid mechanics*, Prentice-Hall, Inc., Englewood Cliffs, N.J., 1962, Reprinted, Dover, New York, 1989.
2. H. Bijl and P. Wesseling, A Numerical Method for the Computation of Compressible Flows with Low Mach Number Regions, to appear in *Proceedings ECCOMAS 96*, 1996, P. Le Tallec and J. Périaux (eds.), Wiley, Chichester.
3. S. Eidelman, P. Colella and R.P. Shreeve, Application of the Godunov method and its Second-Order Extension to Cascade Flow Modelling, *AIAA J.*, 22: 1609–1615, 1984.
4. F.H. Harlow and J.E. Welch, Numerical calculation of time-dependent viscous incompressible flow of fluid with a free surface, *The Physics of Fluids*, 8:2182–2189, 1965.
5. A. Segal, P. Wesseling, J. Van Kan, C.W. Oosterlee and K. Kassels, Invariant Discretization of the Incompressible Navier-Stokes equations in Boundary Fitted Coordinates, *Int. J. Num. Meth. Fluids*, 15: 411–426, 1992.
6. P. van Beek, R.R.P. van Nooyen and P. Wesseling, Accurate discretization on non-uniform curvilinear staggered grids, *J. Comp. Phys.*, 117: 364–367, 1995.
7. J.J.I.M. Van Kan, A Second-order accurate pressure correction method for viscous incompressible flow, *SIAM J. Sci. Stat. Comp.*, 7: 870–891, 1986.
8. C. Vuik, Fast iterative solvers for the discretized incompressible Navier-Stokes equations, *Int. J. for Num. Meth. Fluids*, 22: 195–210, 1996.
9. P. Wesseling, P. van Beek and R.R.P. van Nooyen, Aspects of non-smoothness in flow computations, in *Computational Methods in Water Resources X*, 1994, A. Peters, G. Wittum, B. Herrling, U. Meissner, C.A. Brebbia, W.G. Gray and G.F. Pinder (eds.), 1263 – 1271, Kluwer, Dordrecht.