A Computational Tool for Analyzing Strong Viscous-Inviscid Interactions in Gasdynamics

BARRY KOREN

Centre for Mathematics and Computer Science P.O. Box 4079, 1009 AB Amsterdam, The Netherlands.

1 Computational Tool

An accurate and efficient numerical method for the steady, compressible Navier-Stokes equations is applied for computing strong viscous-inviscid interactions.

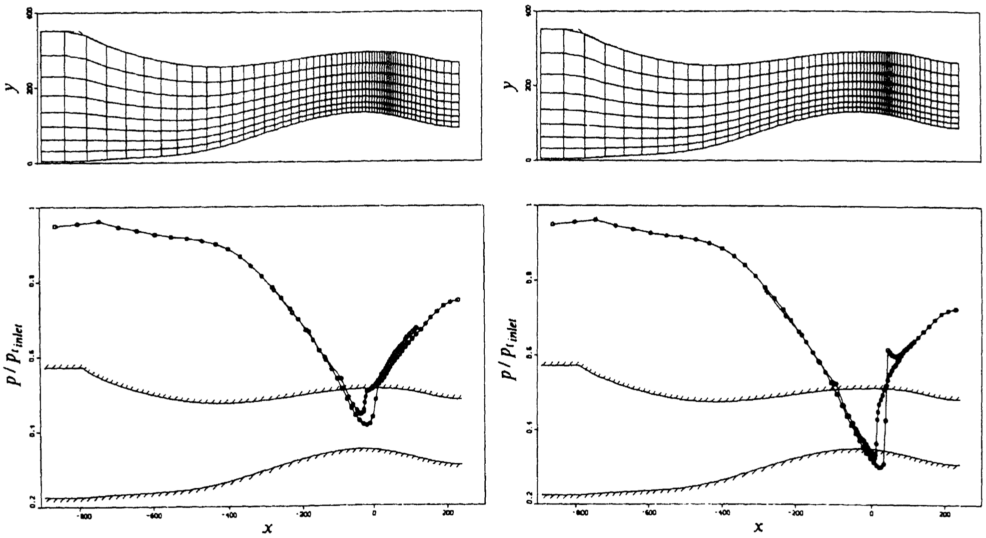
The Navier-Stokes method uses a cell-centred finite volume discretization. For the evaluation of the convective fluxes an approximate Riemann solver is applied. For the diffusive fluxes a second-order accurate central discretization is applied. The solution method requires the flux functions to be continuously differentiable. Well-known approximate Riemann solvers satisfying this requirement are Van Leer's [4] and Osher's [7]. It has been shown that for reasons of accuracy Osher's scheme is to be preferred above Van Leer's. The convective discretization is first-order accurate by taking the left and right Riemann state equal to that in the corresponding adjacent volume. Higher-order accuracy is obtained by applying higher-order piecewise polynomial state interpolation. A limiter is used which allows monotone, third-order accurate flux evaluations. For further details on the discretization we refer to [2].

Nonlinear multigrid is applied for an efficient solution of the system of discretized Navier-Stokes equations. Collective symmetric point Gauss-Seidel relaxation is applied as the smoother. The solution method is very efficient for the first-order discretized Navier-Stokes equations. Difficulties arising when applying the solution method to higher-order discretized equations are circumvented by introducing iterative defect correction as an outer iteration. For further details on the solution method we refer to [3].

An important property of the numerical method is that it does not have an upper bound in the Reynolds number (Re) above which it cannot be applied. The method is hybrid in the sense that it allows the computation of both viscous ($1/Re > 0$) and inviscid ($1/Re = 0$) flows. This property appears to be very useful for the study of viscous-inviscid interactions.

2 Transonic Shock Wave - Boundary Layer Interaction

An important physical feature for the design of transonic air foils is the interaction between the possible shock wave(s) at the air foil and the boundary layers along the air foil. In transonic aerodynamics a lot of work, both experimental and theoretical, is devoted to this so-called transonic shock wave - boundary layer interaction.



a. $M_u = 1.15$. b. $M_u = 1.37$.

Figure 1: 56 × 8-grids and lower surface - pressure distributions, o: computed (first-order), •: measured.

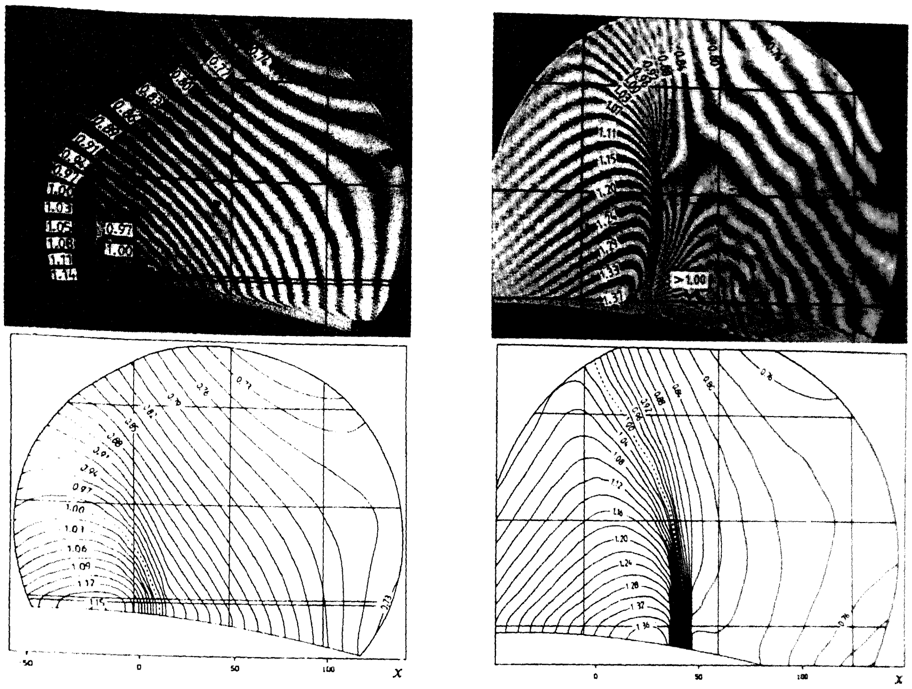
At the Delft University of Technology, a transonic wind tunnel section has been designed and constructed [5] for performing measurements on this phenomenon [6]. Limited accessibility to the flow in this wind tunnel section inhibits measurements throughout the entire flow field. However, knowledge of the entire flow field is important for redesign purposes. This situation motivated a computation of the entire flow field. As a suitable flow model the steady, two-dimensional Euler equations have been chosen (i.e. the steady two-dimensional Navier-Stokes equations with $1/Re = 0$).

In the following, for a choked and non-choked flow, a comparison is made between the Mach number distributions computed with the Euler code and those obtained by wind tunnel experiments.

Grids applied and lower surface - pressure distributions obtained for $M_u = 1.15$ and $M_u = 1.37$, M_u being the Mach number just upstream of the shock wave, are shown in Fig. 1. A very satisfactory agreement is found away from the shock wave. Yet, an important result as the pressure rise across the shock wave at the wall is also predicted in a satisfactory way. (The latter indicates that the Euler code may be exploited for designing experimental set-ups like this indeed.)

In Fig. 2, details of computed Mach number distributions are compared with those obtained (in the wind tunnel) by holographic interferometry. It appears that the computational and experimental results show a nice quantitative agreement away from the wall and shock wave.

The differences between computational and experimental results now can also



a. $M_u = 1.15$. b. $M_u = 1.37$.

Figure 2: Interferometric and computed (first-order) Mach number distributions.

be exploited. Given an Euler code which has proved to be reliable, its results can be considered confidently as experimental results with viscosity and heat conduction switched off. They can be used for identifying simple viscous phenomena and, in particular, complicated viscous-inviscid phenomena.

3 Supersonic Shock Wave - Boundary Layer Interaction

The supersonic interaction to be computed is the well-known experiment for oblique shock wave - boundary layer interaction as considered by Hakkinen et als. at $Re = 2.96\ 10^5$ [1]. The most delicate flow feature in this problem is a shock induced separation followed by a re-attachment; a viscous-inviscid interaction.

The grid applied is optimized for convection by a stretching in flow direction and, in particular, by alignment with the impinging shock wave, leading to an oblique grid (Fig. 3a). A grid adaptation for diffusion is realized by a stretching in cross flow direction.

The limited [2] Navier-Stokes solution shows an acceptable qualitative agreement with the measured results (Fig. 3b). Here, the corresponding Euler flow solution is also computed (Fig. 3c). By doing this, insight is given about the amount of false diffusion present in the viscous solution. The fact that the present method can

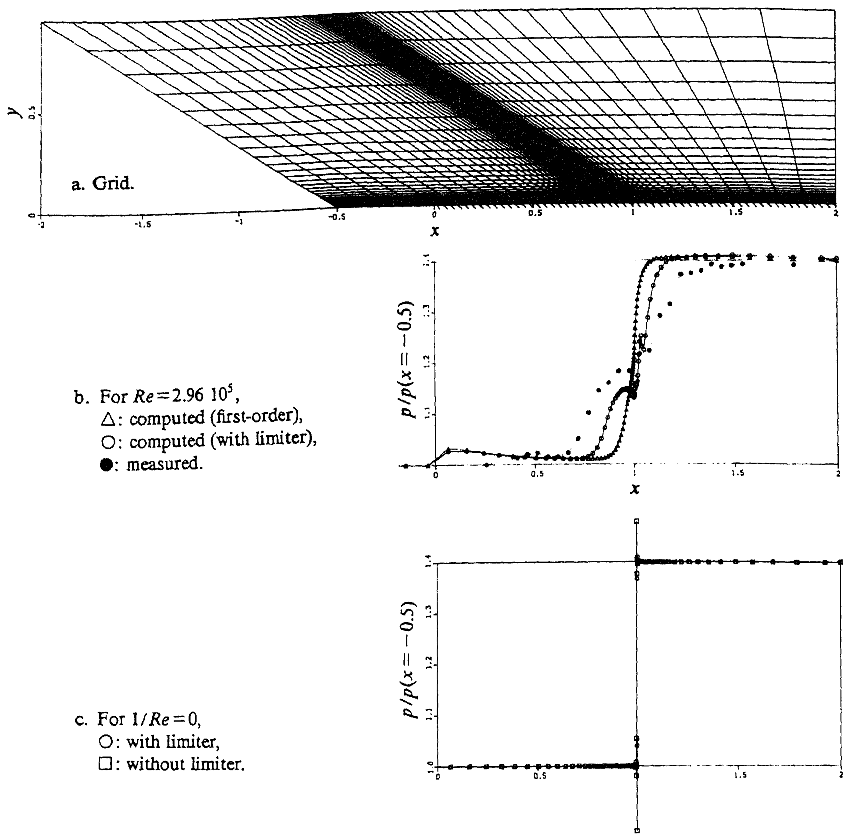


Figure 3: Surface pressure distributions on oblique 80 × 32-grid.

be used for both viscous and inviscid flows makes this comparison easy. Making the comparison is important. For example, when applying for this flow problem a commonly used rectangular grid (Fig. 4a), a viscous surface pressure distribution is obtained which seems to be very close to the measured data (Fig. 4b). However, the corresponding inviscid distribution indicates that this good resemblance is mainly caused by false numerical diffusion in the discretization of the convective terms (Fig. 4c), and therefore is deceptive.

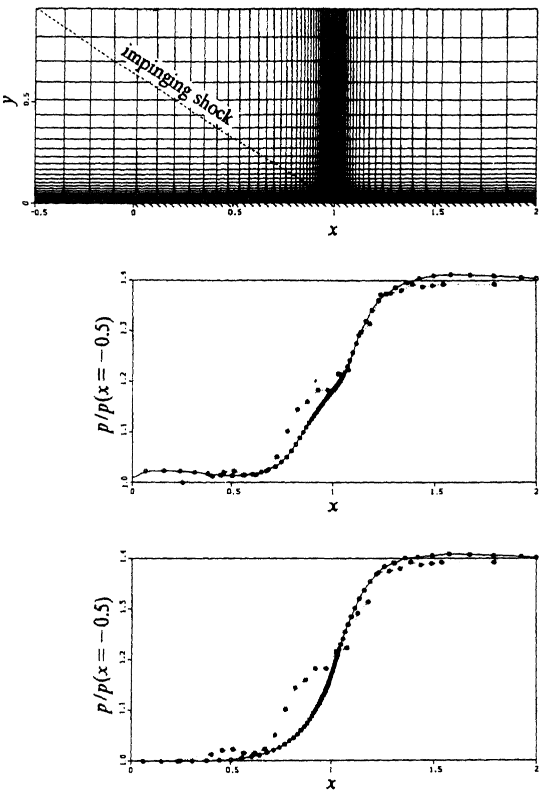
References

[1] R.J. Hakkinen, I. Greber, L. Trilling and S.S. Abarbanel, *The Interaction of an Oblique Shock Wave with a Laminar Boundary Layer*, NASA, Washington, Memo 2-18-59 W (unpublished, 1959).

[2] B. Koren, Int. J. Numer. Meth. in Fluids **11** (1990) 99-117.

[3] B. Koren, J. Comput. Phys. **87** (1990) 25-46.

[4] B. van Leer, *Flux-Vector Splitting for the Euler Equations*, in Proceedings of the Eighth International Conference on Numerical Methods in Fluid Dynamics, Aachen,



a. Grid.

b. For $Re = 2.96\ 10^5$,
O: computed (with limiter),
●: measured.

c. For $1/Re = 0$,
O: computed (with limiter),
●: measured.

Figure 4: Surface pressure distributions on rectangular 80×32-grid.

1982, Lecture Notes in Physics **170** 507-512, edited by E. Krause (Springer, Berlin, 1982).

[5] C. Nebbeling and W.J. Bannink, *A Curved Test Section for Research on Transonic Shock Wave - Boundary Layer Interaction*, Delft University of Technology, LR-414 (unpublished, 1984).

[6] Nebbeling and B. Koren, *Transonic Shock Wave - Turbulent Boundary Layer Interaction on a Curved Surface*, Delft Progr. Rep. **12** (1988) 365-380.

[7] S. Osher and F. Solomon, Math. Comput. **38** (1982) 339-374.