NOTES ON
NUMERICAL FLUID MECHANICS

VOIUME 18

Marie Odile Bristeau
Roland Glowinski
Jacques Periaux
Henri Viviand (Eds.)

Numerical Simulation of Compressible Navier-Stokes Flows

Marie Odile Bristeau
Roland Glowinski
Jacques Periaux
Henri Viviand (Eds.)

Numerical Simulation of Compressible Navier-Stokes Flows

A GAMM-Workshop

V

Friedr. Vieweg & Sohn Braunschweig / Wiesbaden

CIP-Kurztitelaufnahme der Deutschen Bibliothek

Numerical simulation of compressible Navier-Stokes flows: a GAMM workshop / Marie Odile Bristeau ... (eds.). – Braunschweig; Wiesbaden: Vieweg, 1987.
(Notes on numerical fluid mechanics; Vol. 18)
ISBN 3-528-08092-2

NE: Bristeau, Marie Odile [Hrsg.]; Gesellschaft für Angewandte Mathematik und Mechanik; GT

Manuscripts should have well over 100 pages. As they will be reproduced photomechanically they should be typed with utmost care on special stationary which will be supplied on request. In print, the size will be reduced linearly to approximately 75 %. Figures and diagrams should be lettered accordingly so as to produce letters not smaller than 2 mm in print. The same is valid for handwritten formulae. Manuscripts (in English) or proposals should be sent to the general editor Prof. Dr. E. H. Hirschel, Herzog-Heinrich-Weg 6, D-8011 Zorneding.

The addresses of the editors of the series are given on the last page.

Produced by W. Langelüddecke, Braunschweig
Printed in Germany

ISSN 0179-9614

ISBN 3-528-08092-2

Notes on Numerical Fluid Mechanics
Volume 18

Volume 1 Boundary Algorithms for Multidimensional Inviscid Hyperbolic Flows (K. Förster, Ed.)

Volume 2 Proceedings of the Third GAMM-Conference on Numerical Methods in Fluid Mechanics (E. H. Hirschel, Ed.) (out of print)

Volume 3 Numerical Methods for the Computation of Inviscid Transonic Flows with Shock Waves (A. Rizzi / H. Viviand, Eds.)

Volume 4 Shear Flow in Surface-Oriented Coordinates (E. H. Hirschel / W. Kordulla)

Volume 5 Proceedings of the Fourth GAMM-Conference on Numerical Methods in Fluid Mechanics (H. Viviand, Ed.) (out of print)

Volume 6 Numerical Methods in Laminar Flame Propagation (N. Peters / J. Warnatz, Eds.)

Volume 7 Proceedings of the Fifth GAMM-Conference on Numerical Methods in Fluid Mechanics (M. Pandolfi / R. Piva, Eds.)

Volume 8 Vectorization of Computer Programs with Applications to Computational Fluid Dynamics (W. Gentzsch)

Volume 9 Analysis of Laminar Flow over a Backward Facing Step (Ken Morgan / J. Periaux / F. Thomasset, Eds.)

Volume 10 Efficient Solutions of Elliptic Systems (W. Hackbusch, Ed.)

Volume 11 Advances in Multi-Grid Methods (D. Braess / W. Hackbusch / U. Trottenberg, Eds.)

Volume 12 The Efficient Use of Vector Computers with Emphasis on Computational Fluid Dynamics (W. Schönauer / W. Gentzsch, Eds.)

Volume 13 Proceedings of the Sixth GAMM-Conference on Numerical Methods in Fluid Mechanics (D. Rues / W. Kordulla, Eds.) (out of print)

Volume 14 Finite Approximations in Fluid Mechanics (E. H. Hirschel, Ed.)

Volume 15 Direct and Large Eddy Simulation of Turbulence (U. Schumann / R. Friedrich, Eds.)

Volume 16 Numerical Techniques in Continuum Mechanics (W. Hackbusch / K. Witsch, Eds.)

Volume 17 Research in Numerical Fluid Dynamics (P. Wesseling, Ed.)

Volume 18 Numerical Simulation of Compressible Navier-Stokes Flows (M. O. Bristeau / R. Glowinski / J. Periaux / H. Viviand, Eds.)

PREFACE

With the advent of super computers during the last ten years, the numerical simulation of viscous fluid flows modeled by the Navier-Stokes equations is becoming a most useful tool in Aircraft and Engine Design. In fact, compressible Navier-Stokes solvers tend to constitute the basic tools for many industrial applications occuring in the simulation of very complex turbulent and combustion phenomena. In Aerospace Engineering, as an example, their mathematical modelization requires reliable and robust methods for solving very stiff non linear partial differential equations.

For the above reasons, it was clear that a workshop on this topic would be of interest for the CFD community in order to compare accuracy and efficiency of Navier-Stokes solvers on selected external and internal flow problems using different numerical approaches.

The workshop was held on 4-6 December 1985 at Nice, France and organized by INRIA with the sponsorship of the GAMM Committee on Numerical Methods in Fluid Mechanics.

The organizers wish to express their thanks to :

- Professor U. Schumann, Chairman of the GAMM Committee for giving them the possibility to hold this workshop,
- Professor R. Peyret for presenting a survey paper and for fruitful discussions during the preparation of the workshop,
- Professor J. Allegre, Drs M. Raffin and J.C. Lengrand for providing experimental results which give an added interest to the workshop,
- chaipersons for their active parts in directing open discussions,
- all the attendees for their effort in complying with the requirements on the presentation of results,
- Mrs O. Labbé who efficiently contributed to the comparison of the results of Problem (B) by carrying out the plotting of the numerical data,
- the "Service des Relations Extérieures" at INRIA whose help contributed for a large part to the successof the meeting,
- Mrs C. Barny and C. Dubois for their careful typing during the preparation of the workshop and of the synthesis.

M.O. BRISTEAU, INRIA

R. GLOWINSKI, Univ. of HOUSTON/INRIA

J. PERIAUX, AMD/BA

H. VIVIAND, ONERA.

Addresses of the editors of the series „Notes on Numerical Fluid Mechanics":

Prof. Dr. *Ernst Heinrich Hirschel* (general editor)
Herzog-Heinrich-Weg 6
D-8011 Zorneding
FRG

Prof. Dr. *Keith William Morton*
Oxford University Computing Laboratory
Numerical Analysis Group
8-11 Keble Road
Oxford OX1 3QD
Great Britain

Prof. Dr. *Earll M. Murman*
Department of Aeronautics and Astronautics
Massachusetts Institut of Technology (M.I.T.)
Cambridge, MA 02139
U.S.A.

Prof. Dr. *Maurizio Pandolfi*
Dipartimento di Ingegneria Aeronautica e Spaziale
Polytecnico di Torino
Corso Duca Degli Abruzzi, 24
I-10129 Torino
Italy

Prof. Dr. *Arthur Rizzi*
FFA Stockholm
Box 11021
S-16111 Bromma 11
Sweden

Dr. *Bernard Roux*
Institut de Mécanique des Fluides
Laboratoire Associé au C.R.N.S. LA 03
1, Rue Honnorat
F-13003 Marseille
France

CONTENTS

PRESENTATION OF PROBLEMS AND DISCUSSION OF RESULTS

M.O. BRISTEAU
INRIA, B.P. 105, 78153 LE CHESNAY CEDEX, FRANCE
R. GLOWINSKI
UNIVERSITY OF HOUSTON, TEXAS, U.S.A. and INRIA
J. PERIAUX
AMD/BA, B.P. 300, 92214 SAINT CLOUD, FRANCE
H. VIVIAND
ONERA, 92320 CHATILLON, FRANCE

1. INTRODUCTION

The workshop was organized with the specific purpose to bring together a small number of scientists highly concerned with the numerical solution of the compressible Navier-Stokes equations.

Two test problems namely external flows past an airfoil (A) and internal flows in a double throat nozzle (B) were defined with the following features :

i) they should be on simple analytical geometries, suited for a wide range of methods (finite differences, finite volumes, finite elements, spectral, etc...).

ii) they should be 2-D laminar viscous flows with moderate gradients in order to avoid the introduction of turbulent models and also large instabilities due to strong shock-boundary layer interactions, too much costly in CPU time for researchers with limited computer facilities.

In practice the proposed test cases correspond to transonic or supersonic steady flows at low or moderate Reynolds numbers. Two tests cases for problem (A) were chosen because of existing experimental results.

For both problems (A) and (B) the main challenge was the computation of the pressure and skin friction coefficients on the wall.

2. THE PROBLEMS FOR ANALYSIS

2.1. Fluid characteristics

For both problems (A) and (B), we assume that

- the fluid is a perfect gas with constant specific heats C_p, C_v, of ratio :

$$\gamma = C_p / C_v = 1.4$$

- the viscosity coefficient λ, μ verify the Stokes relation :

$$3\lambda + 2\mu = 0$$

- the viscosity coefficients and the thermal conductivity coefficient k are constant and the Prandtl number is :

$$Pr = \mu C_p / k = 0.72$$

2.2. Problem (A) : External 2D flow around a NACA0012 airfoil

Geometry

We consider the NACA0012 airfoil extended to a closed trailing-edge (with zero thickness). It is a symmetrical airfoil and the equation of the upper surface is :

$$\tilde{y}(\tilde{x}) = 5\, t(0.2969\, \tilde{x}^{1/2} - 0.126\, \tilde{x} - 0.3516\, \tilde{x}^2 + 0.2843\, \tilde{x}^3 - 0.1015\, \tilde{x}^4),$$

where

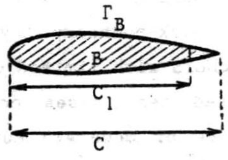
$\tilde{x}$ = the chordwise distance from the leading edge at $\tilde{x} = 0$ referred to c_1,

$\tilde{y}$ = the upper-surface coordinate referred to c_1,

$t = 0.12$ (value of thickness parameter for NACA0012).

For the closed profile

$$0 \le \tilde{x} \le 1.008930411365$$



$$c = 1.0089\, c_1$$

Taking the chord c of the closed profile as reference length, the dimensionless coordinates to be used are :

$$x = \frac{\tilde{x}}{1.0089} \quad , \quad y = \frac{\tilde{y}}{1.0089}$$

Note : The extended profile has a relative thickness of 0.12/1.0089.

Boundary Conditions

The following boundary conditions are prescribed :

at Γ_B,

$\underset{\sim}{u}_B = \underset{\sim}{0}$, adherence condition

$T_B = T_0 = T_\infty \left(1 + \frac{\gamma-1}{2} M_\infty^2\right)$, free stream total temperature ;

at infinity, a uniform flow defined by the following parameters :

M_∞ = Mach number at infinity,

α = angle of attack,

Re = Reynolds number, $Re = \frac{|\underset{\sim}{u}_\infty| \rho_\infty c}{\mu}$.

Test Cases

Seven test cases are proposed :

A1 :	M_∞ = .8	α = 10°	Re = 73 (mandatory)
A2 :	M_∞ = .8	α = 10°	Re = 500 (mandatory)
A3 :	M_∞ = 2.	α = 10°	Re = 106 (mandatory)
A4 :	M_∞ = 2.	α = 10°	Re = 1000 (optional)
A5 :	M_∞ = .85	α = 0°	Re = 500 (mandatory)
A6 :	M_∞ = .85	α = 0°	Re = 2000 (optional)
A7 :	M_∞ = .85	α = 0°	Re = 10000 (optional)

The six cases A_1-A_6 give steady flows, only the optional case A_7 is associated with an unsteady flow.

The cases A_1 and A_3 have been prescribed because experimental results were available for these data of the problem. It would have been interesting to associate slip conditions to these cases because they are closer to the conditions of the experiments.

The test case A_2 leads to a separated flow. The main feature of the supersonic cases A_3 and A_4 is a detached bow shock.

The Reynolds numbers of case A_5, A_6, A_7 associated with the same Mach number are chosen in order to allow the comparison of the thickness of the boundary layers and to check that they vary as $1/\sqrt{Re}$.

Numerical results

The authors have selected the most representative results among the following numerical outputs of the proposed test cases :

- Plots of the mesh,
- Wall distributions :

  a) pressure coefficient, $C_p = \dfrac{p - p_\infty}{\frac{1}{2}\rho_\infty |\underset{\sim}{u}_\infty|^2}$,

  b) skin friction coefficient, $C_f = \dfrac{\tau_w}{\frac{1}{2}\rho_\infty |\underset{\sim}{u}_\infty|^2}$,
  (τ_w = wall shear stress)

  c) heat flux coefficient, $C_h = \dfrac{q}{\frac{1}{2}\rho_\infty |\underset{\sim}{u}_\infty|^3}$

  (q = heat flux, positive if directed from the wall to the fluid)

- Contour maps

  a) Mach number,
  b) pressure,
  c) density.

- Convergence history plots of the following residuals :

$$\frac{1}{\Delta t} \frac{\|\rho^{n+1} - \rho^n\|_{L^2}}{\|\rho_\infty\|_{L^2}},$$

$$\frac{1}{\Delta t} \frac{\|\rho \underset{\sim}{u}^{n+1} - \rho \underset{\sim}{u}^n\|_{L^2}}{\|\rho_\infty \underset{\sim}{u}_\infty\|_{L^2}},$$

$$\frac{1}{\Delta t} \frac{\|e^{n+1} - e^n\|_{L^2}}{\|\rho_\infty |\underset{\sim}{u}_\infty|^2\|_{L^2}}.$$

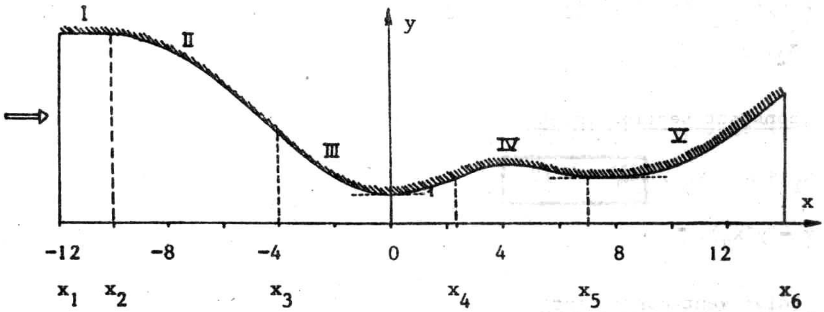
2.3. Problem (B) : Internal flow in a double throat nozzle

The double throat nozzle of test problem (B) was designed with the aim of generating strong viscous interaction phenomena in steady, laminar, compressible flows in a well-bounded domain. Supersonic flow conditions are first obtained by means of a simple converging-diverging nozzle. Then the wall is turned concave and a converging channel region is thus formed. It is in this diverging-converging part of the nozzle, with partly supersonic flow conditions, that we expect compression waves, shock waves and separation phenomena to occur. Then the flow goes through a second throat and is allowed to expand rapidly in a second diverging channel.

Geometry

This plane symmetrical nozzle is shown on the figure below. The wall is made up of 5 polynomial arcs, with continuity of slope and of curvature, except for the points $x = x_2$ and $x = x_5$ where the curvature is discontinuous.

The origin of the abscissa, $x = 0$, is taken at the first throat. The half-height at $x = 0$ is chosen as reference length in the following equations.



Nozzle geometry

Arc III (first throat region)

$x_3 \le x \le x_4$, $\boxed{x_3 = -4,\ x_4 = 2.3}$

$$y = 1 + \frac{a}{2} x^2 \left[\frac{x^2}{6} - \frac{x_3+x_4}{3} x + x_3 x_4 \right] , \quad \boxed{a = -0.03}$$

We deduce :

$$y(x_3) = 1 + \frac{a}{6} x_3^3 (2x_4 - \frac{1}{2} x_3)$$

$$y'(x_3) = \frac{a}{2} x_3^2 (x_4 - \frac{1}{3} x_3)$$

$$y(x_4) = 1 + \frac{a}{3} x_4^3 (x_3 - \frac{1}{4} x_4)$$

$$y'(x_4) = \frac{a}{2} x_4^2 (x_3 - \frac{1}{3} x_4)$$

The points $x = x_3$ and $x = x_4$ are inflexion points ($y" = 0$). The throat radius of curvature is $R = \frac{1}{a \, x_3 \, x_4} = \frac{1}{0.276}$.

Arc II (convergent)

$$x_2 \le x \le x_3 \quad , \quad \boxed{x_2 = -10}$$

$$y = y(x_3) + y'(x_3)(x - x_3) \left[1 - \frac{1}{3} \left(\frac{x - x_3}{x_2 - x_3}\right)^2\right]$$

We deduce :

$$y(x_2) = y(x_3) + \frac{2}{3} (x_2 - x_3) \, y'(x_3)$$

At $x = x_2$, $y' = 0$.

Arc I (constant section inlet)

$$x_1 \le x \le x_2 \; , \quad \boxed{x_1 = -12.}$$

$$y = y(x_2).$$

Arc IV (divergent-convergent)

$$x_4 \le x \le x_5 \; , \quad \boxed{x_5 = 7.}$$

The second throat is at $x = x_5$, $y'(x_5) = 0$, with $y(x_5)$ given :

$$\boxed{y(x_5) = 1.6}$$

$$y = y(x_4) + (x-x_4)\ [y'(x_4) + A_5\ X^2 + B_5\ X^3]$$
$$X = (x-x_4)\ /\ (x_5-x_4)$$

$$A_5 = 4\ C_5 - 3\ y'(x_4)$$
$$B_5 = -3\ C_5 + 2\ y'(x_4)$$
$$C_5 = (y(x_5) - y(x_4))\ /\ (x_5-x_4)\ .$$

We deduce

$$y''(x_5 - 0) = \frac{6}{x_5 - x_4}\ [-2\ C_5 + y'(x_4)] > 0$$

Arc V (divergent)

$$x_5 \leq x \leq x_6 \qquad \boxed{x_6 = 14.}$$

$$\boxed{y(x_6) = 5.85}$$

$$y = y(x_5) + [y(x_6) - y(x_5)]\ z^3\ [2-z]$$
$$z = (x-x_5)\ /\ (x_6-x_5)$$

Note that $y'' = 0$ at $x = x_5 + 0$ and at $x = x_6$.

Discussion of problem (B)

Uniqueness of steady solutions of the compressible Navier-Stokes equations for this problem remains an open question from a mathematical point of view, and special attention must be given to the geometry and to some of the flow boundary conditions in order to come as close as possible to a well-posed problem. Since we do not have a complete mathematical proof that this will be the case, we must rely upon the physical understanding of the flow problem. In this respect, it can be noted that an internal flow bounded by inflow and outflow section appears to raise more questions than the unbounded flow past a finite obstacle since, in this latter case, uniform flow conditions apply at infinity (of course practical difficulties may arise because the outer boundary is located at a finite distance).

In order to define unambiguous reference quantities, it is best to assume that the incoming flow at the entrance section possesses an inviscid isentropic core, so that the corresponding reservoir conditions provide the needed reference values (total enthalpy or temperature, sound speed, pressure...). Furthermore, to avoid the problem of the choice of the flow conditions to be specified across a viscous layer at the inflow boundary, we take zero boundary layer thickness at this boundary, which amounts to assuming inviscid isentropic flow across the entire entrance section, except at the wall itself where to no-slip condition is imposed. In fact, whether or not the no-slip condition is imposed at the entrance section does not make any appreciable difference if this section is large enough (compared to throat section) : for example, when the Mach number changes from $M_1 = 0$ to $M_2 \neq 0$ at constant total enthalpy and constant pressure, the relative density variation is $(\rho_2-\rho_1)/\rho_1 = \frac{\gamma-1}{2} M_2^2$, which gives 2×10^{-3} at $M_2 = 0.1$. Moreover this alternative is more or less equivalent to a variation in the location of the nozzle entrance section equal to one mesh size Δx, which is negligible.

Thus entropy and total enthalpy are known over the entire entrance section. If we assume that inviscid flow theory can be applied at this section, then an additional condition is needed in subsonic flow ; the upstream part of the nozzle is made with parallel walls, so that it is reasonable to assume parallel flow ($v = 0$) at the entrance section.

At the exit section, the flow is in a state of rapid expansion, with a large supersonic part, which makes it reasonable to assume that upstream influence (i.e. the elliptic nature of the equations), can be neglected i.e. that the flow conditions in this section are entirely determined from the flow properties upstream without the need to impose some boundary conditions.

Finally, taking into account all the above conditions, we assume that, for given reservoir conditions and given physical reference length (hence given Reynolds number), as in inviscid flow there is a maximum value Q_M of the mass flow rate Q such that no solution exists for $Q > Q_M$ and the solution exists and is unique for $Q = Q_M$. Problem (B) is concerned with this solution corresponding to the maximum mass flow rate which is unknown.

The second throat has been chosen large enough compared to the first one so that the maximum mass flow rate corresponds to a change from subsonic to supersonic flow across the first throat at least in the core around the nozzle axis. In other words the solution for maximum mass flow rate is determined by the first throat.

Boundary conditions

At the wall

- no-slip condition $\underset{\sim}{u}_W = \underset{\sim}{0}$
- temperature equal to the reservoir temperature : $T_W = T_0$

Upstream boundary (inlet section, $x = x_1 = -12.$)

- total enthalpy equal to reservoir enthalpy, i.e. :

$$\frac{\gamma}{\gamma-1}\frac{p}{\rho} + \frac{1}{2}\underset{\sim}{u}^2 = \frac{\gamma}{\gamma-1}\frac{p_0}{\rho_0} = \frac{a_0^2}{\gamma-1} \qquad (1)$$

- entropy equal to reservoir entropy, i.e. :

$$p\,\rho^{-\gamma} = p_0\,\rho_0^{-\gamma} \qquad (2)$$

An additional condition may be specified, but all the flow properties cannot be known in advance since the mass flow rate is not arbitrary. A natural condition is that of parallel flow, i.e. : $v = 0$, where v is the y-component of velocity.

Test Cases

The only remaining parameter is the Reynolds number defined as :

$$Re_0 = a_0\, L\, \rho_0 / \mu$$

where the subscript 0 refers to reservoir conditions, and L is the nozzle half-height at the first throat $x = 0$. (we recall that the viscosity is a constant).

The flow in this nozzle should be computed for the following three values of the Reynolds number :

test case B_1 : $Re_0 = 100$
test case B_2 : $Re_0 = 400$
test case B_3 : $Re_0 = 1600$

Numerical results

The suggested numerical outputs were the following :

- Plots of the mesh,
- Wall distributions :
  a) pressure , p/p_0
  b) skin friction coefficient, $C_f = \dfrac{\tau_w}{\frac{1}{2}\rho_0\, a_0^2}$,
  (τ_w = wall shear stress)