NUMERICAL AND PHYSICAL ASPECTS OF AERODYNAMIC FLOWS II

Edited by Tuncer Cebeci

Numerical and Physical Aspects of Aerodynamic Flows II

Edited by Tuncer Cebeci

With 171 Illustrations

Springer-Verlag
New York Berlin Heidelberg Tokyo

Tuncer Cebeci
Douglas Aircraft Company
3855 Lakewood Boulevard
Long Beach, CA 90846
U.S.A.

and

California State University
Long Beach, CA 90840
U.S.A.

Library of Congress Cataloging in Publication Data
Symposium of Numerical and Physical Aspects of
Aerodynamic Flows (2nd : 1983 : California State
University, Long Beach)
Symposium of Numerical and Physical Aspects of
Aerodynamic Flows II.
Includes index.
1. Aerodynamics—Congresses. I. Cebeci, Tuncer.
TL570.S95 1983 629.132'32 83-12482

Typeset by Science Typographers, Medford, NY.
Printed and bound by R. R. Donnelley & Sons, Harrisonburg, VA.
Printed in the United States of America.

9 8 7 6 5 4 3 2 1

ISBN 0-387-12659-7 Springer-Verlag New York Berlin Heidelberg Tokyo
ISBN 3-540-12659-7 Springer-Verlag Berlin Heidelberg New York Tokyo

Professor KEITH STEWARTSON, D.Sc., F.R.S.
1925–1983

This volume is dedicated to Keith Stewartson in recognition of his considerable contributions to fluid dynamics in general and to the subject of this book in particular. He is sadly missed by his friends and colleagues.

Preface

The Second Symposium on Numerical and Physical Aspects of Aerodynamic Flows was held at California State University, Long Beach, from 17 to 20 January 1983. Forty-eight papers were presented, including Keynote Lectures by A. M. O. Smith and J. N. Nielsen, in ten technical sessions which were supplemented and complemented by two Open Forum Sessions, involving a further sixteen technical presentations and a Panel Discussion on the "Identification of priorities for the development of calculation methods for aerodynamic bodies." The Symposium was attended by 120 research workers from nine countries and, as in the First Symposium, provided a basis for research workers to communicate, to assess the present status of the subject and to formulate priorities for the future. In contrast to the First Symposium, the papers and discussion were focused more clearly on the subject of flows involving the interaction between viscous and inviscid regions and the calculation of pressure, velocity and temperature characteristics as a function of geometry, angle of attack and Mach number. Rather more than half the papers were concerned with two-dimensional configurations and the remainder with wings, missiles and ships.

This volume presents a selection of the papers concerned with two-dimensional flows and a review article specially prepared to provide essential background information and link the topics of the individual papers. The decision to concentrate on two-dimensional flows was taken because the related papers provided the best compromise between a cohesive pattern of research activity and the economy of space so necessary in published volumes of proceedings. The papers concerned with three-dimensional flows are in the unpublished volume prepared by the University and available from the Chairman of the Mechanical Engineering Department. Those presented here have been modified and improved as a consequence of suggestions made by the session chairmen and the authors themselves. In most cases, they have been shortened to meet the stringent limitations imposed by the need to ensure the best value for each printed page. It is a pleasure to acknowledge the willingness with which the contributors undertook the task of modification and the efforts which they so successfully made to ensure that their papers conveyed essential information, and thereby contributed to the archival nature of the volume.

The review article, prepared by Professors K. Stewartson and J. H. Whitelaw and myself, attempts to put the individual papers in context with each other and with the larger body of information available in recent literature. It also provides recommendations for future research which are based, in part, on the related Panel Discussion of the Symposium at which statements were provided by L. Keel, R. E. Melnik, H. McDonald, M. W. Rubin and H. Yoshihara. The Keynote paper by A. M. O. Smith is concerned with wings, as well as airfoils, and provides a practical perspective for those whose current research emphasis is on two-dimensional flows. The remaining 22 papers have been placed in four parts which bring together the closely related papers and provide a convenient framework for the reader.

The Symposium was made possible partly by financial support provided to the California State University by NASA Ames and Langley Research Centers, the U.S. Army Research Office (ARO), the National Science Foundation (NSF) and the Naval Sea Systems Command (NAVC), and also by the cooperation of authors, session chairmen, participants and colleagues at the University. Particular thanks are due to W. F. Ballhaus and V. L. Peterson of NASA Ames, D. Bushnell of NASA Langley, R. E. Singleton of ARO, G. K. Lea of NSF, L. Pasiuk of NAVSEA and H. Unt of the University. The content of the volume was decided after extensive discussions, especially with D. Bushnell, H. McDonald, K. Stewartson and J. H. Whitelaw. The editing process benefitted considerably from the efforts of Nancy Barela and Sue Schimke and it is a pleasure to acknowledge their help.

Long Beach, California — TUNCER CEBECI
April 1983

Contents

PART 2. Interactive Methods and Measurements: Subsonic Flows

PART 3. Interactive Methods: Transonic Flows

Calculation of Two-Dimensional Flow Past Airfoils

T. Cebeci,* K. Stewartson,† and J. H. Whitelaw‡

1. Introduction

The classical method of calculating flows past airfoils is to assume that viscous effects are confined to their immediate neighborhood and that the remainder of the flow-field can be assumed inviscid. Hence the pressure distribution may be computed from potential-flow equations and the drag subsequently obtained from the turbulent boundary-layer equations. This procedure has a considerable range of practicability but is inappropriate for the solution of some practical problems which occur, especially at high angles of attack and with the more complicated airfoil shapes in use at the present time. At low speeds these problems are chiefly associated with separation, both for laminar and turbulent flows, and with the understanding of the correct representation of transition and turbulent flow processes. At transonic speeds the appearance of shocks in the inviscid part of the flow field and their interaction with the boundary layer represent additional difficulties.

A principal handicap faced by the older schools of scientists in this area was the lack of computers to test the mathematical models of the flow field that had been suggested and to guide theoreticians towards the development

*Department of Mechanical Engineering, California State University, Long Beach, CA 90840.

†Department of Mathematics, University College, London WCIE 6BT

‡Department of Mechanical Engineering, Imperial College, London SW7 2BX.

of more appropriate ones. The situation changed during the 1960s as computer technology helped scientists to provide better insight into flowfields. For many, the critical point was reached with the paper by Murman and Cole [1] on inviscid transonic flow with shocks; afterwards the computer revolution in aerodynamics was underway. An illustration of the change is that whereas in 1970 less than 5% of papers in the AIAA Journal were devoted to computations, in 1980 it was 23% [2].

In this review we consider recent advances in our abilities to calculate subsonic and transonic flows around practical airfoils and in our understanding of their nature. The review, and the individual papers which follow, should be viewed in the context of similar efforts with the three-dimensional flows associated with wings and the design methods presently in use in the aircraft industry. The volume of papers presented at the Second Symposium [3] shows that the techniques developed for two-dimensional flows are readily extended to represent three-dimensional flows although the design process is still conducted with sophisticated potential-flow methods and simple viscous methods which are frequently based on the solution of integral equations. Further information on the procedures in current use by aircraft companies may be found, for example, in the papers of Lynch [4, 5], Bristow [6], Gilmer and Bristow [7], and Wigton and Yoshihara [3].

The emphasis of the review, and of the papers which follow it, is on calculation methods and their ability to represent à priori the flow around airfoils as a function of shape, angle of attack and Mach number. The following section provides a brief general review of the constituent components of calculation methods and in particular describes different forms of the conservation equations and their relative merits, together with the implications of assumptions which may be invoked to permit their numerical solution and to represent the important characteristics of transitional and turbulent flows. Particular procedures which have been used to represent airfoil flows are considered in Section 3 which evaluates and compares procedures involving the interaction of solutions of the potential-flow and boundary-layer equations and the solution of time-averaged forms of the Navier–Stokes equations with a solution domain which encompasses the potential-flow region.

As a consequence of this emphasis, little is said about experimental techniques although their successful use is essential to the evaluation and improvement of the calculation methods. Three of the papers in this volume, i.e., those of Render and Stollery, Adair, Thompson and Whitelaw, and Nakayama, present measurements of pressure and velocity characteristics obtained by techniques which include pressure transducers, hot-wire anemometry, flying-wire anemometry and laser velocimetry. These papers, and the references which they contain, allow an interested reader to pursue these topics further. The volume edited by Emrich [8] and particularly Chapter 9, will provide a more general introduction. It must be emphasized that all of the calculation methods discussed in Sections 2 and 3 and in the following papers are approximate and that the achievement of a truly predictive calculation method requires the incorporation of physical information and the use of numerical solution methods. The former is essentially based on experimental evidence of the particular flow phenomenon in question and the latter, although based on knowledge of the numerical assumptions, requires evaluation with specific boundary conditions and again experiments are essential. Summary conclusions are presented in Section 4 together with recommendations based, in part, on a panel discussion held at the Symposium to consider priorities for future developments.

2. Reynolds-Averaged Navier–Stokes Equations and Their Solution

It is generally accepted, with a very great deal of a posteriori justification but nevertheless partly as an act of faith, that the unsteady Navier–Stokes equations for a compressible fluid together with a suitable equation of state, often that for a perfect gas, are sufficient to describe the flow past a practical airplane. Unlike simpler versions they are free from paradox [9] and while the mathematical theory is not yet complete, sufficient progress has been achieved to give us confidence that solutions do exist with the general properties experience tells us they must have [10]. Their only fundamental weaknesses are that they do not appear to describe flow details in the interior of moderate or strong shocks and at the tip of a very sharp edge. Neither weakness is material here. Even at a Mach number of 1.05 and a Reynolds number of 10^7 the shock thickness is about the same as that of the laminar sublayer of a turbulent boundary layer [11] and as Mach number rises the thickness rapidly decreases. Thus no significant loss in accuracy arises by regarding it as an unresolved discontinuity, conditions on either side being related by the Rankine–Hugoniot conditions, about the validity of which there appears to be no dispute. A difficulty may arise with the shock, fitted in this way, as it penetrates the boundary layer since the discontinuity changes to a region of rapid variation and eventually the shock loses its identity. However, at the present stage of the development of the computational art the shock is captured as a finite transition region and the effect of the boundary layer is to modify its structure and thickness. The failure at the tip of a sharp edge is local and confined to distances small compared with the radius of curvature of the nose of a practical airfoil.

Even so, the full numerical solution of these equations is not seriously considered as a reasonable goal of computational effort at the present time due to the difficulty of resolving all the scales of the motion especially in regions of turbulent flow. The principal effort is devoted to the solution of reduced forms of the Reynolds-averaged Navier–Stokes equations in which the resolution of the turbulence is limited, for example, to a grid dimension of the order of $\delta/20$, where δ represents the corresponding characteristic dimension of the flow, and the effects of the subgrid turbulence are modelled. The various flow properties are averaged over the grid element and also over a time much shorter than that believed to be of interest and we obtain the equations:

$$\frac{\partial \rho}{\partial t} + \frac{\partial \rho u_i}{\partial x_i} = 0 \tag{1}$$

$$\rho \frac{\partial u_i}{\partial t} + \frac{\partial}{\partial x_i}(\rho u_i u_j) = -\frac{\partial p}{\partial x_i} + \frac{\partial \sigma_{ij}}{\partial x_j} \tag{2}$$

$$\rho \frac{\partial h}{\partial t} + \frac{\partial}{\partial x_j}(\rho h u_j) = u_j \frac{\partial p}{\partial x_j} + \frac{\partial p}{\partial t} - \frac{\partial q_j}{\partial x_j} + \sigma_{ij}\frac{\partial u_i}{\partial x_j} \tag{3}$$

The coordinates in these equations are Cartesian, p, ρ are the averaged pressure and density, and u_i, h are the velocity components and enthalpy-averaged with a mass-weighting [12]. Further σ_{ij} and q_j denote the un-weighted averaged stress tensor and heat flux with

$$\sigma_{ij} = \mu\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} - \frac{1}{3}\delta_{ij}\frac{\partial u_k}{\partial x_k}\right) - R_{ij} \tag{4}$$

$$q_j = \frac{\mu}{\Pr}\frac{\partial h}{\partial x_j} + Q_j \tag{5}$$

where μ is the viscosity so that $\nu = \mu/\rho$ is the kinematic viscosity.

$$R_{ij} = \langle \hat{\rho} u_i' u_j' \rangle, \qquad Q_j = \langle \hat{\rho} u_j' h' \rangle \tag{6}$$

$\langle\ \rangle$ denotes the averaged value, the caret means the actual value before averaging, and the prime denotes the difference between the actual value and the mass-weighted average. It should be noted that the mass-weighted forms of dependent variables, for example $u_i \equiv \langle \hat{\rho}\hat{u}_i \rangle / \rho$, are equal to the unweighted forms for Mach number less than around 0.3 and that the difference is of little practical significance at Mach numbers less than around 0.5. At higher Mach numbers, and where comparison with measurement is required, it is important to know whether the unweighted or weighted property has been calculated or measured. The bulk viscosity is ignored and although it is not necessarily zero for gases with polyatomic molecules such as air, its effect is only significant in shock interiors which have been excluded from consideration.

The adaptation of these equations to a form suitable for computation requires three principal steps which are the subjects of the following subsections. First, the form of equation depends on presumed physical features of the flow under consideration: for example, the equations required to represent a low-speed attached boundary layer are simpler than those required for a high-speed flow with a large region of separation. Secondly, the numerical procedure required to solve the equations depends on the form of equations and its successful implementation depends on the method of discretization within the solution domain. Thirdly, the terms R_{ij} and Q_j must be expressed in terms of dependent or independent variables. These steps are considered here in a general way and, in the following sections, in relation to specific investigations.

2.1. Some Reduced Forms of the Navier–Stokes Equations

If the viscous and conduction terms are omitted from the equations of motion (1) to (3), they reduce to Euler's equations for inviscid flow. The solution of these equations is the ultimate goal of inviscid flow studies and at the present time methods based on their time-dependent forms are under intense development but few specific applications to practical airfoils have been made. Among relevant papers are two studies by Jameson and Schmidt, one with Turkel [13] in which they obtain converged solutions for a symmetric airfoil in transonic flow and another with Whitfield and Thomas, in this volume, in which the effects of viscosity are added.

More commonly a further approximation is made by assuming that the flow is irrotational when the governing equations take on the potential form

$$\begin{aligned}(a^2-u^2)\frac{\partial^2\phi}{\partial x^2}-2uv\frac{\partial^2\phi}{\partial x\,\partial y}+(a^2-v^2)\frac{\partial^2\phi}{\partial y^2}\\ -2u\frac{\partial^2\phi}{\partial x\,\partial t}-2v\frac{\partial^2\phi}{\partial y\,\partial t}-\frac{\partial^2\phi}{\partial t^2}=0\end{aligned} \tag{7}$$

where (u, v) are the velocity components, ϕ the potential and a the velocity of sound. If the flow is incompressible, ϕ satisfies Laplace's equation and there are a variety of methods available for its determination, the most common being the so-called panel method which uses both a source and a dipole or vorticity density on surface panels that represent the body about which flow is to be computed [6, 14]. Conformal mapping techniques employing rapidly convergent iteration schemes and highly efficient numerical procedures (including the Fast Fourier transform) are also useful and have enabled complex multielement airfoil systems to be studied [15, 16]. Once compressibility effects are significant, commonly when $M > 0.4$, the use of such distributions on the airfoil is generally inappropriate. However, if the significant compressibility effects are limited in extent, the nonlinearities in Eq. (7) may be interpreted as source distributions in the flow field to be computed iteratively from the source distribution on the airfoil. Otherwise finite-difference or finite-element methods must be used. For wholly subsonic flow these methods prove to be quite satisfactory; with the Kutta assumption at the trailing edge and provided separation does not occur in the boundary layer, they can by themselves give lift coefficients within 10% of the true values. The associated drag is zero, however, since the effects of viscosity are ignored.

For transonic flow, two additional problems arise. First, Eq. (7) changes its character and becomes hyperbolic in the supersonic part of the flowfield.

Second, shocks can appear in this part and cannot be completely resolved by a solution of Eq. (7) for the assumption that the flow is potential is violated behind curved shocks, from which vorticity is generated and travels downstream along the broad streamlines. Moreover Eq. (7) assumes that the entropy is constant everywhere so that p/ρ^{γ} is invariant. In fact the entropy jumps across a shock although the jump is only $O(\varepsilon^3)$ when the shock strength is $O(\varepsilon)$. The general opinion appears to be that the equations are best solved in conservative form as a set of first-order equations of the type

$$\frac{\partial Q}{\partial t} + \frac{\partial C_i}{\partial x_i} = 0 \tag{8}$$

[Eq. (1) is an example] which facilitates the capture of discontinuities and avoids the generation of fictitious sources along them. In addition, it is easier to satisfy the requirements of mass conservation.

A simplified version of these equations is the transonic small perturbation equation (TSP), which follows from Eq. (7) by assuming that $u \simeq a$ and $v \ll a$, and is relevant to flow past thin airfoils at small angles of attack lying close to the x-axis. The steady form of the TSP equation is

$$\left[1 - M_\infty^2 - (\gamma+1) M_\infty^2 \phi_x\right] \phi_{xx} + \phi_{yy} = 0 \tag{9}$$

A number of equivalent forms have been used in practice in an attempt to overcome some small but systematic discrepancies. Cheng and Meng [17] have pointed out that one source of the discrepancies may be removed by an improved description of the critical speed on the body. This leads to a redefinition of the transonic similarity parameter now close to that proposed by Murman and Cole. Integration through the sonic point has caused problems in the past but recently new schemes have been proposed to overcome them by ensuring the entropy must increase through a shock transition, Enquist and Osher [18]. The TSP equation particularly as implemented by the LTRAN2 computer code has proved to be very valuable in design studies of practical airfoils, both in steady and unsteady motion and the achievements to date are described in an excellent review by Ballhaus et al [19].

There are, however, frequently significant discrepancies between theory and experiment which can only be explained by viscous effects. We have already mentioned errors in lift and drag for incompressible flow and these grow progressively larger as the Mach number increases even when the flow remains attached. Indeed at transonic speeds viscous effects can reduce the lift by 50% and move the shock by 20%–30% of chord [20]. The classical way of overcoming this deficiency is to introduce the boundary-layer concept. The usual form of the boundary-layer equation is

$$u\frac{\partial u}{\partial s} + v\frac{\partial u}{\partial n} = -\frac{1}{\rho}\frac{dp}{ds} + \frac{1}{\rho}\frac{\partial}{\partial n}\left(\mu\frac{\partial u}{\partial n}\right) - \frac{\partial}{\partial n}(\langle u'v'\rangle) \tag{10}$$

where s, n denote distances along and normal to the airfoil or the wake line and u, v are the corresponding velocity components. The position of the wake-line is assumed to be fixed to a first approximation by inviscid theory as is the pressure distribution along it and on the airfoil surface. In a hierarchical approach to the calculation of the flow properties, the inviscid equations are solved first and then the boundary-layer equations. This procedure is unsatisfactory near the trailing edge where the boundary layer exerts a significant influence on the external flow and at separation where it leads to a singularity [21] and hence a contradiction. It is more usual to adopt an interactive approach where the inviscid and boundary-layer equations are solved simultaneously. The solutions of the boundary-layer equations provide us with the displacement thickness δ^* and the momentum thickness θ on the airfoil and in the wake and these may be used as boundary conditions for the inviscid flow calculations which then give the pressure distribution needed for Eq. (9) and the wake curvature κ. This procedure is sometimes referred to as the "standard problem" as opposed to the "inverse problem" where the wall shear stress or displacement thickness is used as an extra boundary condition for the boundary-layer equations. These boundary-layer properties may be implemented by defining a displacement airfoil, which surrounds and extends to infinity downstream of the rigid airfoil but is not closed, together with a given pressure jump across it, or by defining a blowing velocity from the surface of the airfoil and a velocity discontinuity across the streamline issuing from the trailing edge (wake-line). The two approaches are equivalent, each having the same theoretical range of validity but the second has the advantage that only the boundary conditions need changing as the iteration proceeds, the grid remaining largely unaltered. More extended accounts of these procedures are given by Veldman [22] and by Lock [23].

For laminar flow $\langle u'v' \rangle$ is zero and the solution of Eq. (7), together with the corresponding forms of the continuity and energy equations, is possible without the addition of physical information other than boundary conditions. The solution of Eq. (10), however, requires that the velocity correlation $\langle u'v' \rangle$ be specified in terms of dependent or independent variables and this topic is discussed in Section 2.3. In situations where compressibility or heat transfer is important, the energy equation must also be solved and turbulent flow again requires knowledge of a correlation term, in this case $\langle v'h' \rangle$.

The interactive scheme is effective for a wide class of practical airfoils but is inappropriate when there are substantial regions of separated flow over the airfoil except when used semi-empirically (see Section 3). Moreover the cross-stream pressure gradient becomes important when the curvature of the airfoil surface is large or when the boundary layer is thick. One obvious way is to incorporate a second momentum equation of the form

$$\frac{\partial p}{\partial n} = \kappa \rho u^2 \tag{11}$$

where κ is the curvature of the airfoil surface. Strictly, this equation is limited to boundary layers that are thin since for thick layers the curvature of the streamline varies significantly across them. Huang, Groves and Belt [24] point out that good results may then be obtained by replacing κ in Eq. (11) by some mean value. In such circumstances, however, the sharp distinction between the boundary layer and the inviscid flow becomes less valid and it is natural to look for ways in which the two sets of equations can be combined into one without having to invoke the full equations. One possibility is to add the viscous terms of the boundary layer equations, $\partial/\partial n[\mu(\partial u/\partial n)]$ and $\partial/\partial n\langle u'v'\rangle$, to the inviscid equations to obtain a set sometimes referred to as the parabolized Navier–Stokes equations and used by Rubin and Reddy in this volume. These equations may then be solved by a combination of the methods appropriate to the inviscid and boundary-layer equations separately. Thus for two-dimensional incompressible laminar flow over a flat plate lying along the x-axis, we have, in the simplest form,

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} = 0$$

$$\frac{\partial u}{\partial t} + u\frac{\partial u}{\partial x} + v\frac{\partial u}{\partial y} = -\frac{1}{\rho}\frac{\partial p}{\partial x} + \nu\frac{\partial^2 u}{\partial y^2} \tag{12}$$

$$\frac{\partial v}{\partial t} + u\frac{\partial v}{\partial x} + v\frac{\partial v}{\partial y} = -\frac{1}{\rho}\frac{\partial p}{\partial y}$$

Near the trailing-edge formal objections can be lodged against the boundary-layer concept but for laminar flow the interactive approach does at least provide the principal features of the flow [25]. If, as is usually the case, the flow is turbulent there, the assumption $\partial/\partial n \gg \partial/\partial s$ fails [26] but nevertheless a considerable measure of success has been achieved in using the concept to predict the gross and important features of the flowfield, e.g. Butter and Williams [27] for incompressible flows and LeBalleur [28] who also studied transonic flows. Melnik [20] has examined the structure of the turbulent boundary layer near the trailing edge on the assumption that $\partial/\partial n \sim \partial/\partial s$ and the turbulence is frozen.

The boundary-layer concept, used in the interactive mode, can provide a good representation of small regions of separation on the airfoil. When they occupy a significant portion, the concept becomes inadequate and it is generally accepted that recourse must be had to a solution of the full Reynolds averaged Navier–Stokes equations. It should be recognized that the size and strength of recirculation which requires the solution of these equations remains to be quantitified and the merits or representing cross-stream pressure gradients by different forms of the y-momentum equation are also imprecisely defined. Solutions of the Navier–Stokes equations obtained to date have a mixed record of success.