Computers and Fluids 126 (2016) 102–128

Contents lists available at ScienceDirect

Computers and Fluids

journal homepage: www.elsevier.com/locate/compfluid

Steady-state laminar flow solutions for NACA 0012 airfoil

R.C. Swanson [a,*], S. Langer [b]

[a] NASA Langley Research Center, Computational AeroSciences Branch, Hampton, VA 23681, United States
[b] DLR, Deutsches Zentrum für Luft- und Raumfahrt, Lilienthalplatz 7, Braunschweig D-38108 , Germany

ARTICLE INFO

Article history:
Received 18 August 2015
Revised 21 November 2015
Accepted 24 November 2015
Available online 4 December 2015

Keywords:
Navier–Stokes
Steady state
Laminar
Runge–Kutta
Preconditioner

ABSTRACT

In this paper we consider the solution of the compressible Navier–Stokes equations for a class of laminar airfoil flows. The principal objective of this paper is to demonstrate that members of this class of laminar flows have steady-state solutions. These laminar airfoil flow cases are often used to evaluate accuracy, stability and convergence of numerical solution algorithms for the Navier–Stokes equations. In recent years such flows have also been used as test cases for high-order numerical schemes. While generally consistent steady-state solutions have been obtained for these flows using higher order schemes, a number of results have been published with various solutions, including unsteady ones. We demonstrate with two different numerical methods and a range of meshes with a maximum density that exceeds 8×10^6 grid points that steady-state solutions are obtained. Furthermore, numerical evidence is presented that even when solving the equations with an unsteady algorithm, one obtains steady-state solutions.

1. Introduction

In the development and evaluation of numerical schemes it is essential to have a comprehensive set of test problems. Such problems can allow specific properties of the solution algorithm to be validated. Furthermore, they can possibly delineate deficiencies in the algorithm that must be overcome. These test cases should provide either an analytic solution or a fully converged and resolved numerical solution. While experimental data can often provide confirmation for the validity of the computed solution, one should not necessarily conclude that everything is working correctly for a steady-state problem just because there is reasonable agreement with the data according to some engineering criterion (such as a small variation in the aerodynamic coefficients) when the residual has only been reduced a few orders of magnitude (rather than machine zero). An appropriate evaluation should involve a hierarchy of test cases. For example, if we consider solving the Navier–Stokes equations, the test cases should include laminar flows. Laminar flow test cases have the distinct advantage that they eliminate the need at sufficiently high Reynolds numbers to either resolve turbulence in a flow field or to model the turbulence appropriately.

Some possible laminar flow test cases involving the NACA 0012 airfoil were considered by the first author of this paper in 1984. One of these test cases (Mach number (M) of 0.5, angle of attack (α) of 0°, and Reynolds number (Re) equal to 5000) was introduced in the 1985 paper by Swanson and Turkel [1] to evaluate an algorithm for solving the compressible Navier–Stokes equations. This particular case was chosen because it has a small amount of trailing edge separation (beginning at approximately the 81% chord location), and thus, represented a good way to check the levels of dissipation being produced by the numerical scheme. For example, if the scheme is too dissipative, the effective Reynolds number is reduced, and the separation point moves downstream. Since that time, this case has been considered in numerous evaluations of schemes, such as Refs. [1–7]. It has also been used in a study of vortex–airfoil interaction by Svärd et al. [8]. In addition, this case and another one of the original cases ($\alpha = 3^\circ$) have also been used by Venditti [9] in evaluating a grid adaptive scheme for functional outputs of flow simulations.

In recent years some additional laminar flow cases for the NACA 0012 airfoil have also been considered. The flow conditions are the same as for the original cases of Swanson except $\alpha = 1^\circ$ or $\alpha = 2^\circ$. These cases, as well as the original cases, have been considered in the European project ADIGMA to develop adaptive high-order variational methods for aerospace applications [10]. Due to the strong interest in these laminar flow cases, there is a need to have documentation of their solutions.

Some recent efforts to compute members of this class of flow problems have produced solutions that are inconsistent with the rather large number of previous solutions (computed with a wide range of numerical methods), including those already cited in this paper. For example, Abgrall and De Santis [11] show unexpected solutions for the $\alpha = 0^\circ$ case, which should have a symmetric solution

* Corresponding author. Tel.: +1 757 864 2235; fax: +1 757 864 8816.
E-mail addresses: r.c.swanson10@gmail.com (R.C. Swanson), Stefan.Langer@dlr.de (S. Langer).

http://dx.doi.org/10.1016/j.compfluid.2015.11.009
0045-7930/

based on previous results, using both second-order and third-order discretizations. One possibility for such results is the presence of an asymmetry in their solution algorithm. Another example concerns the unsteady solution for the $\alpha = 2^\circ$ case presented in the ADIGMA Project [10] paper by Taube et al. (starting on page 427). Even though the computational method included adaptation, the calculation was performed on a rather coarse mesh (815 cells) and the adaptation seems to have been focused in the wake. Thus, there could have been insufficient resolution in the neighborhood of the separation point, which can produce a result that appears to be an unsteady solution. There is also the possibility that the solution algorithm is not sufficiently strong, as measured by the level of implicitness. In this paper we will show how a weak solution algorithm can produce what appears to be an unsteady solution. Dolejší [12] has also obtained an unsteady solution for this laminar flow case. A nonlinear system of algebraic equations (from discontinuous Galerkin finite element method) is solved with an inexact Newton-time method. An inner iterative method is required to solve a linear system at each time level. Since the residual (algebraic error) of the iteration is not being reduced to approximately zero, there can be sufficient error to produce an unsteady result. In any of these examples one cannot dismiss the possible influence of boundary conditions. It is difficult to assess this because descriptions of the discrete implementation of the boundary conditions for the problems being considered are not given.

As a consequence of such examples, a primary objective of this paper is to demonstrate that there are steady-state solutions, even on high density meshes, to this class of problems. Two additional objectives of this paper are as follows. The first of these is to examine and document the behavior of a set of laminar flow problems, which includes those just discussed, as well as a commonly considered lower *Re* case with a higher angle of attack. The second additional objective is to compare two methods for solving the compressible Navier–Stokes equations. One method [13,14] is based on structured grids, and the other [15] is based on unstructured grids. Both methods apply a finite-volume approach for spatial discretization. The structured grid method uses a cell-centered formulation, and the unstructured grid method uses a node-centered formulation. Each method applies a matrix form for numerical dissipation. In addition, using the structured grid approach a Roe dissipation (which is frequently used in numerical schemes) is also considered. When comparing what we call structured and unstructured schemes it is important to clarify the meaning of this terminology with respect to contrasting the schemes. In the paper, the same set of structured grids (i.e., quadrilateral elements) is used for both methods. Thus, the comparison of the schemes is based on the fact that there are differences in their elements and implementation.

For all computations in the paper we solve the full Navier–Stokes equations (i.e., all physical diffusion terms are retained). The solvers for the two methods are comprised of a Runge–Kutta (RK) scheme with an implicit preconditioner (herein also designated as RK/Implicit scheme) to extend stability and allow a large Courant–Friedrichs–Lewy (CFL) number. A multigrid scheme is employed for convergence acceleration. In the comparison of the methods we consider accuracy, stability, and convergence when computing solutions of the laminar flow cases.

In the first section of the paper we present the governing flow equations and the boundary conditions for the continuous problem of flow past an airfoil. The next two sections describe and discuss the structured and unstructured methods, including the discrete boundary conditions, that are considered. After delineating the specific differences in the two methods, we then present a set of laminar flow results that compare the methods. Convergence histories are shown for both steady-state and unsteady computations. Comparisons are made between the results with the structured and unstructured methods. The effects of mesh density on the solutions, which contain between 8192 and about 8.39×10^6 cells, as well as the surface pressure and skin-friction distributions for each laminar flow case are presented and discussed. In all cases there is flow separation, and both the separation location and downstream extent of separation are compared. Overall flow details are also illustrated with Mach and streamline plots. Such details provide a reference for other investigators to compare against.

2. Governing equations

We consider the two-dimensional (2-D) Navier–Stokes equations for compressible flow. Assuming a volume fixed in space and time, the integral form of these equations can be written as

$$\int\int_{\mathcal{V}} \frac{\partial \mathbf{W}}{\partial t} d\mathcal{V} + \int_{\mathcal{S}} \mathcal{F} \cdot \mathbf{n} d\mathcal{S} = 0, \tag{2.1}$$

where $\mathbf{W}$ is the state vector of conservative variables, $\mathcal{F}$ is the flux density tensor, and $\mathcal{V}$, $\mathcal{S}$, and $\mathbf{n}$ denote the volume, surface, and outward facing normal of the control volume. One can split the flux density tensor into a convective contribution $\mathcal{F}_c$ and a viscous contribution $\mathcal{F}_v$, which are given by

$$\mathcal{F}_c = \begin{bmatrix} \rho \mathbf{q} \\ \rho u \mathbf{q} + p \mathbf{e}_x \\ \rho v \mathbf{q} + p \mathbf{e}_y \\ \rho H \mathbf{q} \end{bmatrix}, \qquad \mathcal{F}_v = \begin{bmatrix} 0 \\ \bar{\tau} \cdot \mathbf{e}_x \\ \bar{\tau} \cdot \mathbf{e}_y \\ \bar{\tau} \cdot \mathbf{q} - \mathbf{Q} \end{bmatrix} \tag{2.2}$$

where $\mathbf{q}$ is the velocity vector with Cartesian components (u, v), and the unit vectors $(\mathbf{e}_x, \mathbf{e}_y)$ are associated with the Cartesian coordinates (x, y). The variables ρ, p, H represent density, pressure, and total specific enthalpy, respectively. The stress tensor $\bar{\tau}$ and the heat flux vector $\mathbf{Q}$ are given by

$$\bar{\tau} = \begin{bmatrix} \tau_{xx} & \tau_{xy} \\ \tau_{yx} & \tau_{yy} \end{bmatrix}, \qquad \mathbf{Q} = k \begin{bmatrix} \partial T/\partial x \\ \partial T/\partial y \end{bmatrix} \tag{2.3}$$

with

$$\tau_{xx} = \lambda \left(\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} \right) + 2\mu \frac{\partial u}{\partial x}, \quad \tau_{yy} = \lambda \left(\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} \right) + 2\mu \frac{\partial v}{\partial y},$$
$$\tau_{xy} = \tau_{yx} = \lambda \left(\frac{\partial u}{\partial y} + \frac{\partial v}{\partial x} \right). \tag{2.4}$$

Here, the symbol ∂ indicates partial differentiation, μ and λ are the first and second coefficients of molecular viscosity, k denotes the coefficient of thermal conductivity and T represents the temperature. By the Stokes hypothesis $\lambda = -2/3\mu$, and μ is determined with Sutherland's viscosity law [16]. The thermal conductivity k is evaluated with the constant Prandtl number assumption (i.e., the Prandtl number $Pr = 0.72$).

In order to close the system given by Eq. (2.1) we use the equation of state

$$p = \rho R T \tag{2.5}$$

where R is the specific gas constant.

2.1. Physical boundary conditions

In the continuum case we consider an external flow problem (i.e., flow past a given geometry) on an infinite domain $\Omega \subset \mathbb{R}^2$. Thus, we need to define appropriate conditions at a wall boundary, which we assume to be solid. Later, in the discrete case, we define a finite domain. It will then be necessary to define suitable inflow and outflow boundary conditions.

For viscous flows, the non-penetration condition and the no-slip condition are imposed,

$$\mathbf{q} \cdot \mathbf{n} = 0, \quad \mathbf{q} \cdot \mathbf{t} = 0, \tag{2.6a}$$