ARTICLE IN PRESS

Journal of Computational and Applied Mathematics ▮ (▮▮▮▮) ▮▮▮–▮▮▮

Contents lists available at SciVerse ScienceDirect

Journal of Computational and Applied Mathematics

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

Computational study of incompressible turbulent flows with method of characteristics

Ali Atashbar Orang [a,*], Seyed Esmail Razavi [b], Hamoon Pourmirzaagha [c]

[a] *Department of Mechanical and Aerospace Engineering, Science and Research Branch, Islamic Azad University, Tehran, Iran*

[b] *Faculty of Mechanical Engineering, Tabriz University, Tabriz, Iran*

[c] *Department of Mechanical and Aerospace Engineering, Ramsar Branch, Islamic Azad University, Ramsar, Iran*

HIGHLIGHTS

- We developed a new characteristic-based scheme for incompressible turbulent flows.
- The Navier–Stokes partial differential equations were solved numerically.
- Numerical tests were conducted to flow past a circular cylinder.
- High stability range was achieved which led to fast convergence.
- Accurate results were obtained by applying the new CB method.

ARTICLE INFO

Article history:
Received 26 December 2012
Received in revised form 5 May 2013

Keywords:
Incompressible turbulent flow
Navier–Stokes equations
RANS
Spalart–Allmaras turbulence model
Method of characteristics
Upwind scheme

ABSTRACT

The current study presents numerical investigation of incompressible turbulent flow. Momentum and continuity equations are coupled by the Artificial Compressibility Method (ACM). The one-equation Spalart–Allmaras turbulence model along with RANS (Reynolds Averaged Navier–Stokes) equations is used to calculate Reynolds stresses in 2-D problems. For convective fluxes a characteristic-based (CB) finite-volume solution has been developed. Also, a revised Jameson averaging (AVR) method was implemented. In comparison, the proposed CB upwind scheme, demonstrated very accurate results, high stability and faster convergence. For time discretization, the fifth-order Runge–Kutta scheme is used. As the convergence acceleration techniques, the local time stepping and implicit residual smoothing were applied. Considering flow behavior, suitable boundary conditions have been implemented. Cross flow around a horizontal circular cylinder at high Reynolds numbers has been studied as a benchmark. The results obtained using the new proposed scheme, are in good agreement with the standard available solutions in the literature.

1. Introduction

The most encountered incompressible flows in many industrial cases include fully-developed turbulence. Majority of numerical methods have been developed for compressible flows with decreasing Mach number which because of stiffness of the convective terms in incompressible flows cause convergence and accuracy problems, as mentioned by Volpe [1]. This increases sensitivity to boundary conditions as reported in the works of Gresho [2]. The ACM of Chorin [3] as a pre-conditioner enables the coupling of continuity and momentum equations thus allowing time-marching compressible schemes to be applied for incompressible ones. It has been shown that the ACM requires a fast-converging scheme to satisfy the incompressibility conditions [4]. The CB method for ACM, was proposed by Drikakis et al. [5] based on the definition of primitive

---

[*] Corresponding author.
E-mail address: a.atashbar@srbiau.ac.ir (A. Atashbar Orang).

0377-0427/$ – see front matter
http://dx.doi.org/10.1016/j.cam.2013.05.019

Please cite this article in press as: A. Atashbar Orang, et al., Computational study of incompressible turbulent flows with method of characteristics, Journal of Computational and Applied Mathematics (2013), http://dx.doi.org/10.1016/j.cam.2013.05.019

Nomenclature

$C_{b1}, C_{b2}, C_{v1}, C_{v2}, \sigma, \kappa, C_{w1}, C_{w2}, C_{w3}, C_{t1}, C_{t2}, C_{t3}, C_{t4}$	Constants of the standard Spalart–Allmaras model
CFL	Courant number
$\mathbf{F_N}$	Normal convective flux vector along the x direction
p	Pressure
Re	Reynolds number
R	Viscous flux vector along the x direction
S	Viscous flux vector along the y direction
Ω'	Area of secondary cells
u	Velocity along the x direction
v	Velocity along the y direction
U_∞	Reference velocity
u_N	Normal to face velocity
u_P	Parallel to face velocity
W	Initial variable vector

Greek symbols

α	Pseudo-sound velocity
λ	Wave propagation speed
β	Artificial compressibility
$\tilde{\nu}$	Eddy-viscosity variable
Δt	Time step
ν_T	Turbulent viscosity
ρ	Density
ϕ	Symbol for initial stream variables
Ω	Initial cells' area

variables as functions of their values on the characteristics and formation of compatibility equations. Recently Neofytou [6] proposed revisions in the derivation of the compatibility equations which are used to derive the CB formulas for the calculations of the primitive flow variables within the iterative process which is directly related to the convergence rate. The above references show that the CB scheme along with AC was successfully used in a large number of applications [7]. Recently Razavi and Zamzamian [8] have proposed a multidimensional characteristic-based scheme coupled with ACM. This scheme has been successful in reducing the convergence steps and giving accurate results. Other CB method is the Roe scheme which was originally developed for simulation of compressible Euler equations. Razavi and Atashbar extended the Roe scheme to estimate convective fluxes of incompressible viscous flows past a hydrofoil. This method is based on the propagations of virtual acoustic waves which enables damping the numerical oscillations [9].

Nithiarasu and Liu [10] implemented artificial compressibility to solve incompressible turbulent flows at medium Reynolds numbers with characteristic-based scheme. It has been used for solutions of steady and unsteady flows. They used three different turbulence models namely Wolfenstein l, Spalart–Allmaras and standard $k-\varepsilon$. They showed that the artificial compressibility based on CB is suitable for solutions of both steady and unsteady incompressible turbulent flows at medium Reynolds numbers. The finite-volume method along with hybrid grid and $k-\varepsilon$ turbulence model was proposed by Feng and Shanhong [11]. Hybrid grid can be used for accelerating convergence in the unsteady time step. Yang and Chang [12] used 2-D numerical simulation of turbulent flow fields for the NREL S809 airfoil. They assumed the flow as steady, turbulent and incompressible. The turbulent flow by the revised $k-\varepsilon$ model was adopted to calculate effects of low Reynolds numbers near the wall. Marx [13] implemented the implicit MUSCL scheme to calculate 3-D compressible Navier–Stokes equations. They also described turbulence models for separated flows with emphasis on the non-equilibrium numerical model. Ng et al. [14] developed a finite volume method based on vortex flux flow for compressible flows. In their method, the second-order central finite difference and Runge–Kutta schemes are used for flux and time discretizations, respectively. Pentaris et al. [15] presented predictions for 2-D, steady incompressible flows in laminar and turbulent regimes by the standard $k-\varepsilon$ model. Mass conservation is coupled to other equations by artificial compressibility and pressure correction methods. They also used the second- and the fourth-order artificial dissipation to obtain good convergence and efficiency of $k-\varepsilon$ equations. Erturk et al. [16] used stream function–vorticity equations for steady incompressible Navier–Stokes equations. They solved steady flow inside a driven cavity up to Reynolds numbers 21 000 using a 601×601 fine grid. Their formulation proved to be stable and effective at very high Reynolds numbers and non-orthogonal grids. In addition LES[1] modeling was done

---

[1] Large Eddy Simulation

Please cite this article in press as: A. Atashbar Orang, et al., Computational study of incompressible turbulent flows with method of characteristics, Journal of Computational and Applied Mathematics (2013), http://dx.doi.org/10.1016/j.cam.2013.05.019