Journal of Computational Physics 314 (2016) 414–435

Contents lists available at ScienceDirect

Journal of Computational Physics

www.elsevier.com/locate/jcp

A high-order solver for unsteady incompressible Navier–Stokes equations using the flux reconstruction method on unstructured grids with implicit dual time stepping

Christopher Cox [*,1,2], Chunlei Liang [1], Michael W. Plesniak [2]

Department of Mechanical & Aerospace Engineering, The George Washington University, Washington, DC 20052, USA

ARTICLE INFO

Article history:
Received 12 May 2015
Received in revised form 8 February 2016
Accepted 9 March 2016
Available online 14 March 2016

Keywords:
Navier–Stokes
Artificial compressibility
High-order
Flux reconstruction
Unstructured
Implicit LU-SGS

ABSTRACT

We report development of a high-order compact flux reconstruction method for solving unsteady incompressible flow on unstructured grids with implicit dual time stepping. The method falls under the class of methods now referred to as flux reconstruction/correction procedure via reconstruction. The governing equations employ Chorin's classic artificial compressibility formulation with dual time stepping to solve unsteady flow problems. An implicit non-linear lower–upper symmetric Gauss–Seidel scheme with backward Euler discretization is used to efficiently march the solution in pseudo time, while a second-order backward Euler discretization is used to march in physical time. We verify and validate implementation of the high-order method coupled with our implicit time stepping scheme using both steady and unsteady incompressible flow problems. The current implicit time stepping scheme is proven effective in satisfying the divergence-free constraint on the velocity field in the artificial compressibility formulation within the context of the high-order flux reconstruction method. This compact high-order method is very suitable for parallel computing and can easily be extended to moving and deforming grids.

1. Introduction

In computational fluid dynamics, unstructured high-order methods, i.e. those considered 3rd order and above, are useful for the study of unsteady vortex dominated viscous flows in complex geometries. These methods can provide high accuracy for similar cost as low-order methods [1]. Furthermore, solution acceleration can be achieved with p-adaptivity and p-multigrid methods. However, high-order methods are less robust and more complicated to implement than low-order methods, especially when treating irregular geometries.

Four popular methods have been developed to address the need for high-order accuracy – discontinuous Galerkin (DG), spectral difference (SD), spectral volume (SV) and flux reconstruction/correction procedure via reconstruction (FR/CPR). Discontinuous Galerkin was initially developed for the neutron transport equation by Reed and Hill [2]. The staggered grid spectral method was initially presented by Kopriva [3] and was modified and called spectral difference method by Liu, Vinokur and Wang [4]. The spectral volume method was introduced by Wang, Zhang and Liu [5], where each element is split

* Corresponding author. Tel.: +1 267 567 4375.
E-mail address: ccox@gwmail.gwu.edu (C. Cox).
[1] Computational Aerodynamics & Hydrodynamics Laboratory.
[2] Biofluid Dynamics Laboratory.

http://dx.doi.org/10.1016/j.jcp.2016.03.016

into multiple control volumes. Flux reconstruction was initially developed by Huynh [6,7] and has a close connection to lifting collocation penalty by Wang and Gao [8] (LCP). Consequently, those authors coined the name correction procedure via reconstruction. Further development was made on energy stable flux reconstruction schemes by Vincent, Castonguay and Jameson [9] also known as Vincent–Castonguay–Jameson–Huynh (VCJH) schemes. The FR/CPR methods represent significant development because they have the ability to recover DG, SD and SV for linear problems. Furthermore, they are attractive because of the fact that no surface or volume integrals are required. Also, solution and flux points are arranged in a staggered fashion in SD, while flux points only lie along element interfaces in FR/CPR. As such, the latter method does not require extra flux computations at locations other than interior solution points. Using this fact, Liang, Cox and Plesniak [10] have shown improved computational efficiency of FR/CPR over SD for quadrilateral elements. For a comprehensive review of these methods, see Huynh, Wang and Vincent [11] and Wang [12].

With the development of high-order unstructured methods comes the need to achieve faster convergence, especially for solving large-scale problems using parallel computers. This demand motivates the development of time stepping techniques for which the Courant–Friedrichs–Lewy (CFL) condition is less restrictive, which is hardly the case when explicit (e.g. multi-stage Runge–Kutta) schemes are combined with high-order methods and dual time stepping. In this paper, we present an implicit scheme that overcomes the time step restriction associated with explicit schemes used for solving the unsteady incompressible Navier–Stokes equations. Work done to improve convergence of unsteady incompressible flow can be seen in Liang, Chan and Jameson [13], whereby they use a spectral difference method and Chorin's original artificial compressibility formulation (AC) [14] as well as a p-multigrid method to accelerate the convergence rate of pseudo time stepping for a particular physical time step. However, the p-multigrid method marginally improves the stiffness introduced by the artificial compressibility approach, especially for flows that require high aspect ratio elements near solid walls. As computers become equipped with larger RAM, implicit time stepping schemes are seen as effective drivers to overcome this stiffness. With these implicit schemes much larger time steps can be taken in comparison to explicit schemes, delivering the potential to improve the rate of convergence significantly. Application of the DG method to the incompressible Navier–Stokes equations was performed by Bassi et al. [15], where artificial compressibility was introduced only at the interface flux level. Shahbazi, Fischer and Ethier [16] and Nguyen, Peraire and Cockburn [17] applied DG to these equations as well, using triangular and tetrahedral grids. However, incompressible solvers involving a Poisson solver cannot be easily parallelized according to domain decomposition of the grid. One advantage of the approach taken in our current work lies in the fact that the method is discontinuous and local; as a result, there is no global matrix to split.

In recent years, the lower–upper symmetric Gauss–Seidel (LU-SGS) scheme that was originally developed by Jameson and Yoon [18] with multiple grids for solving the unsteady Euler equations has been used within the high-order CFD community for solving compressible flow problems on unstructured grids using SD [19–21] and SV [22] methods. However, when solving incompressible flows using artificial compressibility, the LU-SGS scheme is more economical because it requires the solution of only three equations in two dimensions as opposed to the four needed for compressible flow. Furthermore, with the introduction of artificial compressibility, pressure and velocity are loosely coupled and the Navier–Stokes equations take on a mixed hyperbolic/parabolic mathematical nature. This loose coupling lends itself to parallel computing as both pressure and velocity are state variables in pseudo time. As such, this paper aims to bring a popular high-order method and time stepping technique for producing high-order accurate solutions for compressible flow to the incompressible flow regime. In this regard the current method is novel, especially if it can be applied to moving and deforming grids needed to solve problems involving fluid–structure interaction (FSI) on massively parallel computers.

2. Governing equations with artificial compressibility

Numerical computation of incompressible flow is challenging because the continuity equation lacks a time-dependent term. To handle this difficulty, consider the two-dimensional unsteady incompressible Navier–Stokes equations with artificial compressibility written in conservation form

$$\frac{\partial \boldsymbol{U}}{\partial \tau} + \boldsymbol{I}_D \frac{\partial \boldsymbol{U}}{\partial t} + \nabla \cdot \boldsymbol{F}(\boldsymbol{U}, \nabla \boldsymbol{U}) = 0 \tag{1}$$

where terms involving τ and t represent pseudo and physical time derivatives, respectively. The vector of state variables $\boldsymbol{U}(x, y, t) \in \Omega$, where $\Omega \subset \mathbb{R}^2$ and $t \geq 0$, and the vector of fluxes $\boldsymbol{F}(\boldsymbol{U}, \nabla \boldsymbol{U})$ are

$$\boldsymbol{U} = \begin{bmatrix} p \\ u \\ v \end{bmatrix}, \qquad \boldsymbol{F}(\boldsymbol{U}, \nabla \boldsymbol{U}) = \begin{bmatrix} \boldsymbol{f} \\ \boldsymbol{g} \end{bmatrix} \tag{2}$$

where $\boldsymbol{I}_D = diag(0, 1, 1)$. The flux vector contains both inviscid and viscous terms in x and y, where $\boldsymbol{f} = \boldsymbol{f}_e - \boldsymbol{f}_v$ and $\boldsymbol{g} = \boldsymbol{g}_e - \boldsymbol{g}_v$. The inviscid fluxes for the artificial compressibility formulation are

$$\boldsymbol{f}_e = \begin{bmatrix} \beta u \\ u^2 + p \\ uv \end{bmatrix}, \qquad \boldsymbol{g}_e = \begin{bmatrix} \beta v \\ uv \\ v^2 + p \end{bmatrix} \tag{3}$$

and the viscous fluxes are