Journal of Computational Physics 276 (2014) 438–467

Contents lists available at ScienceDirect

Journal of Computational Physics

www.elsevier.com/locate/jcp

A high-order compact difference algorithm for half-staggered grids for laminar and turbulent incompressible flows

Artur Tyliszczak

Czestochowa University of Technology, Institute of Thermal Machinery, Al. Armii Krajowej 21, 42-200 Czestochowa, Poland

ARTICLE INFO

Article history:
Received 4 July 2013
Received in revised form 12 June 2014
Accepted 25 July 2014
Available online 1 August 2014

Keywords:
Compact difference scheme
Projection method
Half-staggered mesh

ABSTRACT

The paper presents a novel, efficient and accurate algorithm for laminar and turbulent flow simulations. The spatial discretisation is performed with help of the compact difference schemes (up to 10th order) for collocated and half-staggered grid arrangements. The time integration is performed by a predictor–corrector approach combined with the projection method for pressure–velocity coupling. At this stage a low order discretisation is introduced which considerably decreases the computational costs. It is demonstrated that such approach does not deteriorate the solution accuracy significantly. Following Boersma B.J. [13] the interpolation formulas developed for staggered uniform meshes are used also in the computations with a non-uniform strongly varying nodes distribution. In the proposed formulation of the projection method such interpolation is performed twice. It is shown that it acts implicitly as a high-order low pass filter and therefore the resulting algorithm is very robust. Its accuracy is first demonstrated based on simple 2D and 3D problems: an inviscid vortex advection, a decay of Taylor–Green vortices, a modified lid-driven cavity flow and a dipole–wall interaction. In periodic flow problems (the first two cases) the solution accuracy exhibits the 10th order behaviour, in the latter cases the 3rd and the 4th order is obtained. Robustness of the proposed method in the computations of turbulent flows is demonstrated for two classical cases: a periodic channel with $Re_\tau = 395$ and $Re_\tau = 590$ and a round jet with $Re = 21\,000$. The solutions are obtained without any turbulence model and also without any explicit techniques aiming to stabilise the solution. The results are in a very good agreement with literature DNS and LES data, both the mean and r.m.s. values are predicted correctly.

1. Introduction

High-order discretisation methods are irreplaceable in DNS (Direct Numerical Simulation) and LES (Large Eddy Simulation) studies focused on very deep and detailed analysis of the fluid flow problems. The present status of majority of the high-order methods and possibilities of their applications are presented in a recent review paper [1]. Undoubtedly, from the point of view of the solution accuracy none of discretisation methods may compete with the spectral and pseudo-spectral methods [2] which are regarded the most accurate. Their weak point is that they can be applied in rather simple computational domains and with the nodes distribution and boundary conditions enforced by the applied method. For instance, the spectral approach based on the Fourier series is suited for periodic problems with uniform meshes, the collocation method using the Chebyshev polynomials is defined on the computational points stretched near the boundaries and is suitable for

E-mail address: atyl@imc.pcz.czest.pl.

http://dx.doi.org/10.1016/j.jcp.2014.07.043

wall bounded flows. In both the cases transformations to different node distributions or applications in irregular domains are not trivial and require domain division, normalisation, co-ordinate transformations, etc. From this point of view the high-order compact difference methods [3] seem to be very attractive giving much more possibilities regarding non-uniformity of the computational meshes, selection of the boundary conditions or shapes of computational domains. The most apparent difference between the compact and classical finite difference schemes is the computational stencil. In the compact methods, the approximations of derivatives are obtained by solving linear systems of equations involving all grid points along particular node lines. It makes the approximation global and therefore very sensitive to implementation of the boundary conditions and accuracy of the approximation in regions near boundary domain. Although the compact methods cannot compare with flexibility of the finite volume or finite element based methods, they are successfully applied on non-uniform meshes and in irregular domains, often in combination with domain decomposition approach and with transformation from physical to computational domains [4–7].

In this paper, the high-order compact method (up to 10th order) is applied for incompressible flow problems. One of the main difficulties in simulations of incompressible flow is the calculation of the pressure for which there is no evolution equation nor the equation of state as in the case of simulations of compressible flows. There are number of algorithms developed to determine the pressure field (family of SIMPLE methods, PISO method, projection method, auxiliary potential method and probably more); an overview of existing approaches may be found in [8]. The solution method which is used in this work is based on the projection method [8], where the pressure is obtained from the Poisson equation. The well known problem of the projection method is the pressure oscillation appearing due to decoupling of the velocity and pressure field. A remedy for that is to apply the so-called staggered grids introduced in 60' by Harlow and Welch [9], where the pressure and velocity components are stored in different locations. This is well known approach and is described in books dealing with the fluid flow problems, for instance [8,10,11]. The staggered grid arrangement has been used in low order solvers based on the finite difference or finite volume methods for decades. Recently, the staggered meshes were used also in combination with the high order compact schemes, both for compressible [12,13] and incompressible flows [14–20].

Besides of removing the pressure oscillation the evident advantage of the staggered grids is that the mass conservation is a trivial consequence of the mesh staggering. In case of the standard 2nd order discretisation method of Harlow and Welch [9] the kinetic energy is conserved as well. For higher order methods (e.g. the finite difference 4th order) the momentum and energy conserving discretisation schemes have been also proposed [21]. The conservation properties of staggered arrangement on unstructured meshes were studied in [22]. It was shown that discretisation of the divergence form of the Navier–Stokes equations can conserve both the kinetic energy and momentum.

The staggered grid arrangement has also a few very important disadvantages: (i) it requires interpolation between locations of the velocity components – this increases the computational costs; (ii) at the domain boundaries not all the velocity components are defined explicitly – this leads to difficulties in implementation of the boundary conditions; (iii) for non-uniform and curvilinear meshes the co-ordinate transformation has to be performed at different locations. These problems may be overcome by applying the so-called half-staggered meshes introduced in [23] which also ensure strong coupling between the pressure and velocity field. In this approach all three velocity components are stored in the same locations while the pressure is computed in the points shifted half a cell width from the velocity nodes. Comparing to the fully staggered grid arrangement the half-staggered grids greatly simplify the numerical codes. They facilitate the solutions of the flow problems in complicated domains with almost the same effort as in the case of collocated meshes. Applications of the half-staggered approach on curvilinear meshes were presented in [24,25]. More recently, the half-staggered meshes were used in combination with the immersed boundary approach [26,27] and also for adapting moving meshes [28]. In all these cases the spatial discretisation was performed with the 2nd order schemes. Probably the first application of the half-staggered approach with a higher order discretisation was proposed in [29] using the compact difference schemes. It was shown that separation of the pressure from the velocity locations is sufficient to obtain accurate results with almost non-oscillatory pressure field. The authors proposed an efficient solution method of the Poisson equation through a transformation of the pressure into the spectral space using the Fourier transformation and the cosine Fourier transformation for periodic and symmetric boundaries. Extension of the method to non-periodic cases (for instance wall bounded flows) required introduction of ghost cells. This revealed to be the main factor influencing the accuracy of the proposed method which for a lid-driven flow in a cavity turned out to be only the 2nd order.

In the present work we combine the half-staggered grid arrangement and the compact difference approximation with aim to obtain high order solutions. Nominally the compact methods lead to very small discretisation errors but, as shown in [29], the final results are not always as accurate as one could expect. Apart from accuracy we focus on two aspects: computational efficiency and stability of the proposed algorithm. In particular it is shown that application of low order finite difference schemes for discretisation of the Poisson equation reduces only slightly the overall solution accuracy, but comparing to the fully compact discretisation it gives a significant gain of computational efficiency. Robustness of the proposed method is obtained by a specific way of the velocity interpolation which is unavoidable part of the projection method on staggered meshes. Unlike in [23–25,29] in the present formulation the velocity field is interpolated twice: before and after the pressure correction step. The test cases include periodic, wall bounded and spatially developing flows. It is shown that the solution algorithm is stable and there is no need to use any additional stabilisation methods (filtering, artificial dissipation, etc.). The obtained solutions are stable and accurate in high Reynolds number flows and also in a case of an inviscid flow. The results are completely free from any oscillations in the velocity and pressure field. The order of approximation of the proposed algorithm is first determined based on simple 2D and 3D flow problems for which analytical and reliable