Computers and Fluids 127 (2016) 131–145

Contents lists available at ScienceDirect

Computers and Fluids

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

High-order compact difference algorithm on half-staggered meshes for low Mach number flows

Artur Tyliszczak*

Czestochowa University of Technology, Faculty of Mechanical Engineering and Computer Science, Al. Armii Krajowej 21, 42-201 Czestochowa, Poland

ARTICLE INFO

Article history:
Received 30 March 2015
Revised 2 October 2015
Accepted 23 December 2015
Available online 8 January 2016

Keywords:
Variable density flow
Projection method
Pressure–velocity decoupling

ABSTRACT

The paper presents a solution algorithm for variable density non-isothermal flows based on a high-order compact difference approximation combined with the projection method formulated on half-staggered meshes. In contrast to full-staggered grid arrangement, with the pressure and all scalar variables located in the cell centers, we shift only the pressure nodes while we keep the scalars and velocity components in the same nodes. This greatly simplifies the solution procedure as the interpolation between the nodes takes place only within the projection method. Derivative approximations and interpolation formulas are discretized using the high-order compact difference schemes up to 10th order in the central nodes and third/fourth order schemes near domain boundaries. The proposed algorithm is first verified in natural convection problems in square and rectangular wall bounded cavities for Rayleigh numbers $\mathrm{Ra} = 3.4 \times 10^5$ and $\mathrm{Ra} = 1.0 \times 10^6$ with maximal density ratio equals nine. The results are compared with published data for both steady and unsteady flow regimes. The performed simulations reveal that achievement of accurate results is conditioned by a mesh resolution in thermal boundary layers near the walls, and a way in which the governing equations are formulated, i.e., the conservative or non-conservative form. This turns out more important than a formal order of discretization method. Robustness of the proposed algorithm for the computations of turbulent flow is demonstrated based on the flow in a periodic channel at Reynolds number $\mathrm{Re}_\tau = 200$ with density ratio equal to two. In this case the profiles of mean velocity, temperature and their fluctuations are analyzed and they agree quite well with literature data. In all presented test cases the obtained pressure fields are smooth and without any oscillations observed when the collocated meshes are employed.

1. Introduction

Low Mach number flows with variable density and temperature play very important role in various technological devices, e.g. solar cells, combustion chambers or heat exchangers. Fully or partially wall-bounded geometries are common in these applications as their working principles are based on an exchange of heat between a flowing medium and the walls. Precise numerical solutions of wall bounded flows usually require dense computational meshes and compactness of the grid nodes in the wall vicinities. It is best when such specially prepared meshes are used in combination with high-order approximation methods. From the point of view of the accuracy none of the spatial discretization method may compete with spectral and pseudospectral methods [1]. However, due to imposed distributions of the grid nodes and limitations in prescribing boundary conditions their application is limited to simple academic problems. High-order compact difference methods [2] seem much more attractive from this point of view. Contrary to the spectral methods they allow for easy adaptation of non-uniform meshes and selection of any type of boundary conditions. Concerning irregular domains the compact methods cannot compare with the excellent features of the finite volume or finite element based methods. Nevertheless, more and more literature data show that the irregular domains are not a barrier for the compact methods. In such problems they are implemented through a combination of domain decomposition approach with transformation from the physical to computational domains, see [3–6]. In this paper we limit to simple geometries with computational grids condensed near the walls and we focus on application of the compact difference schemes for solutions of low Mach number flows with large density variations. Two main difficulties in simulations of this type of flows are: (i) lack of computational stability caused by density gradients; (ii) calculation of pressure for which there is no evolution equation nor the equation of state

* Tel.: +48 343250505.
E-mail address: atyl@imc.pcz.czest.pl

http://dx.doi.org/10.1016/j.compfluid.2015.12.014

as in the case of simulations of compressible flows. In this work, based on the projection method [7], we develop an algorithm which is stable for large density ratios and provides accurate results with pressure field without any spurious oscillations.

There are number of algorithms developed to determine the pressure field for incompressible and low Mach number flows, see [7] for an overview of existing approaches, with pressure-correction algorithms such as SIMPLE type methods and projection methods being the most popular. These algorithms can be applied on the so-called collocated or staggered grid arrangements [8]. In the former approach all velocity components, pressure and scalar quantities (e.g. temperature, density, concentration) are stored in the same spatial locations (grid nodes). In the latter, proposed in 1960s by Harlow and Welch [9], the pressure and scalars are shifted from the cell corners to the cell centers and moreover the velocity components are separated and placed on different cell faces. The well known problem in the use of pressure-correction algorithms on collocated meshes is an oscillating (checkerboard) pressure field appearing due to decoupling of the velocity and pressure [8]. Literature aiming to solve this problem is very extensive. In the field of incompressible flows one should mention Rhie and Chow interpolation method [10] which by now is regarded as a cure eliminating the pressure oscillations for low order methods. Its modified variant [11], suitable for unsteady computations has been recently adapted also for low Mach number flows [12,13]. For this class of flows the algorithms based on a pressure-correction through the projection method were studied in details in [14,15]. There, some improvements for flows with sharp density gradients were also proposed. However, these improvements were formulated based on a simple second order spatial discretization method and its extension to high-order approximation methods seems unlikely.

The staggered grid arrangement has been used in low order finite difference and finite volume approaches for decades. The only reason for which it is still being in use seems the same as in 1960s. The staggered grids allow to compute pressure accurately and without oscillations. Besides of that an advantage of the staggered grids is that the mass conservation is a trivial consequence of the mesh staggering. In case of the standard second order discretization method of Harlow and Welch [9] the kinetic energy is conserved as well. Recently, the staggered meshes were used also in combination with the high-order compact schemes, both for compressible [16,17] and incompressible flows [18–24]. Important disadvantages of the staggered grid arrangement are as follows: (i) interpolation between the velocity components increases the computational costs; (ii) not all velocity components can be defined explicitly at the boundaries; (iii) in case of non-uniform and curvilinear meshes the co-ordinate transformation has to be performed at different locations.

The above problems may be overcome applying the so-called half-staggered meshes introduced in [25]. In this approach the velocity components are stored in the same locations while the pressure nodes are located in the cell centers. 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. As shown in [26–32] the half-staggered approach ensures strong coupling between the pressure and velocity field.

In the present paper we combine half-staggered approach for the pressure with the collocated approach for all remaining variables. Unlike as in full-staggered grid arrangement, with the pressure and all scalar variables located in the cell centers, we shift only the pressure nodes while keeping the scalars and velocity components in the same nodes. Interpolation between the pressure nodes and velocity locations follows a specific forward–backward interpolation procedure proposed in [32] where for constant density flows it revealed accurate and having stabilizing effect. Here this method is extended to variable density flows and it turns out that its very good properties are preserved. The proposed algorithm characterizes simplicity typical for the collocated approach and in the same time it shares the best features of the staggered meshes. The algorithm is verified in the computations of steady and unsteady natural convection problems and in the computations of turbulent flow in a channel. In all these cases the solutions are stable, accurate and the pressure field is smooth.

2. Governing equations and solution algorithm

2.1. Low Mach number approximation

In this work we consider a low Mach number flow governed by the so-called low Mach number approximation [33–35] with acoustic modes removed from the solution. In this approach the fluid flow is governed by the continuity equation, the Navier–Stokes equations and the energy equation given as [36,37]:

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

$$\rho\frac{\partial u_i}{\partial t} + \rho u_j\frac{\partial u_i}{\partial x_j} = -\frac{\partial p}{\partial x_i} + \frac{\partial \tau_{ij}}{\partial x_j} + \rho g_i \tag{2}$$

$$\rho C_p\frac{\partial T}{\partial t} + \rho C_p u_j\frac{\partial T}{\partial x_j} = \frac{\partial}{\partial x_j}\left(\kappa\frac{\partial T}{\partial x_j}\right) + \frac{dp_0}{dt} \tag{3}$$

where the viscous stress tensor is defined as:

$$\tau_{ij} = \mu(\partial u_i/\partial x_j + \partial u_j/\partial x_i - 2/3\delta_{ij}\partial u_k/\partial x_k)$$

The symbol ρ stands for the density, u_i—velocity components, p—hydrodynamic pressure, g_i—external force (e.g. gravity), T—temperature, μ—molecular viscosity, κ—heat conductivity, and C_p is the specific heat. The symbol p_0 is the so-called zeroth order pressure interpreted as the thermodynamic pressure connected to the temperature through the equation of state:

$$p_0 = \rho RT \tag{4}$$

where R is the gas constant. The thermodynamic pressure should not be confused with the static hydrodynamic pressure in the Navier–Stokes equations. p_0 is constant in space, and for flows in open domains it is also assumed constant in time, then it follows that $dp_0/dt = 0$. In the closed domains, p_0 is constant in space but it varies in time. From the equation of state (4) we have $p_0/RT = \rho$, which integrated over the flow domain gives:

$$p_0 = \frac{\int_V \rho dV}{\int_V 1/RT dV} = \frac{m_0}{\int_V 1/RT dV} \tag{5}$$

where m_0 is the mass of fluid inside domain.

2.2. Predictor–corrector time integration method

The Eqs. (1)–(3) are integrated in time using a predictor–corrector approach combined with the projection method for pressure–velocity coupling. We follow the work of [37], where this algorithm was verified in the computations of variable density flows using the collocated mesh approach. In the present work we adapt it for the half-staggered meshes presented schematically in Fig. 1. As mentioned in the introduction, one of the main advantages of the present approach over the methods based on the collocated meshes is treatment of the pressure. Apart of the checkboard pattern of the pressure field, there are two main problems when the pressure and velocity nodes are located together. The first one is related to specification of the boundary conditions for the pressure. They are necessary for discretization of the