Computers & Fluids 51 (2011) 60–67

Contents lists available at SciVerse ScienceDirect

Computers & Fluids

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

An auxiliary potential velocity method for incompressible viscous flow

P.K. Papadopoulos

Department of Engineering Sciences, University of Patras, Patras 26504, Greece

ARTICLE INFO

Article history:
Received 2 December 2010
Received in revised form 11 July 2011
Accepted 19 July 2011
Available online 11 August 2011

Keywords:
Auxiliary potential velocity
Incompressible flow
Curved duct
Finite volumes

ABSTRACT

In this article a novel auxiliary potential velocity scheme for incompressible flows is presented. The present method is characterized by high accuracy, robustness and simple implementation. Its advantages are highlighted by applying it to several benchmark problems (internal duct flow, flow over a backward facing step) and by extensive comparison with other numerical methods such as SIMPLE and CVP concerning accuracy and convergence. The accuracy of the predictions of the present method is demonstrated through comparison with experimental data.

1. Introduction

In the past decades, several numerical methodologies have been developed to solve the system of equations that govern incompressible flows. These methodologies can be categorized in several families depending on characteristics, such as the use of primitive variables or not and the handling of the velocity–pressure coupling.

A very broad category includes the methods that use a Poisson equation for the pressure. Some of the most widely used methods of the pressure linked family are the SIMPLE method [1] and its variants SIMPLER [2] and SIMPLEC [3] that employ an iterative procedure for the velocity–pressure coupling. The common characteristic of these methods is that the discretized continuity and momentum equations are combined in order to produce a discrete Poisson equation for the pressure. The handling, however, of the discretized equations can become complicated and cumbersome in flow problems that involve complex geometries. These methods require a staggered grid for the avoidance of spurious oscillations in the pressure. Nevertheless, several techniques have been devised that use cell face interpolations and achieve the suppression of these unphysical oscillations while using collocated grids [4,5].

Another method for incompressible flows is the Continuity–Vorticity–Pressure (CVP) variational equations method [6–8]. The CVP method is very efficient and robust, it does not involve a staggered grid and it produces very accurate results. It is based on the decoupling of the evaluation of the velocity corrections from the pressure correction. However, it involves the solution of three additional equations besides the momentum equations for 2D problems and four additional equations in the 3D case. In that sense, it can be computationally demanding when applied to grids with large numbers of nodes.

Another category of methods for incompressible flows includes the SMAC method [9] and its variants that use an auxiliary potential velocity for the determination of the pressure. Several studies have been published concerning the implementation of the SMAC method to internal and external flows. A comprehensive review of the advances and the implementations of the SMAC method can be found in McKee et al. [10]. Comparisons with other methods such as SIMPLEC and PISO have shown that for unsteady flows the SMAC method is more efficient because it needs less computational effort [11] and that it produces very accurate results, in some cases more accurate than PISO [12]. SMAC is a very efficient method which is based on the auxiliary velocity potential. It uses the fractional step technique, so it is mainly used for transient flows. However, it can be used to predict steady flows by marching in time until the solution no longer changes. In this case a time step is introduced, which is dictated by the stability concern and sometimes it is overly small to be computationally efficient.

Other potential based methods have been used by Briley and McDonald [13] and Fletcher and Bain [14]. The former, study the developing flow in a curved square duct and in their method they separate the velocity in two parts, one for the axial potential flow and one for the secondary flow. Their approach uses a combination of viscous and inviscid flow field, which decomposes the numerical problem by taking into account physical approximations. The potential based method of Fletcher and Bain [14] uses elements from both SMAC and SIMPLE. It involves corrections for the velocity and the pressure field and also the use of an auxiliary potential for the satisfaction of the continuity equation. They use the auxiliary

E-mail address: p.papadopoulos@des.upatras.gr

0045-7930/$ - see front matter © 2011 Elsevier Ltd. All rights reserved.
doi:10.1016/j.compfluid.2011.07.016

potential method for the presentation of an approximate factorization explicit method, suitable for solution on parallel CPUs. They state that the introduction of the auxiliary potential is possible if the velocity correction is assumed irrotational. However, it will be shown in this paper that the velocity correction necessary to satisfy continuity is by definition irrotational and that the auxiliary potential method has general applicability.

The scope of this paper is to present an accurate, robust and computationally efficient, auxiliary potential based method. Its main characteristic is that it uses the auxiliary potential for the evaluation of the velocity corrections and then the pressure correction is obtained from the momentum equation. The auxiliary potential was chosen as a means of computing the corrections in order to achieve easy implementation of the numerical scheme to complex coordinate systems, without the need to discretize the governing equations for the construction of a pressure correction equation. The advantages of the present method are that it can be easily applied to both steady and unsteady flows and that it can produce accurate results with very small computational effort and without a restrictive time step. The new scheme is applied on a cell vertex with pressure in the center (CVPC) grid, thus avoiding the complicated staggered grid. The problem of the spurious oscillations is solved with the use of the CVPC grid without any velocity or pressure interpolations on the cell faces, which are obligatory on collocated meshes, and additionally there is no need for boundary conditions for the pressure. The present method is applied to several benchmark problems and it is compared with experimental data and with other numerical methods such as SIMPLE and CVP for accuracy and computational efficiency.

2. Numerical method

In order to outline the steps of the computational procedure we consider the general case of three-dimensional incompressible flow. We express the governing equations in terms of the non-dimensional variables

$$\boldsymbol{v} = \frac{\boldsymbol{v}'}{\nu/D_h}, p = \frac{p'}{\rho\nu^2/D_h^2}, \boldsymbol{\nabla} = D_h\boldsymbol{\nabla}' \tag{1}$$

where $\boldsymbol{v}'$, p', ρ, ν, D_h are the dimensional velocity, pressure, density, kinematic viscosity and hydraulic diameter respectively. The governing equations are the continuity and momentum equations. We shall describe the present scheme for the steady state case. Thus, the time derivative will be omitted. The extension of the scheme to transient flows is straightforward, if the time derivative is retained in the momentum equation.

Using the lagging of the coefficient technique to linearize the momentum equation, the governing equations at the $n+1$ iteration become

$$\boldsymbol{\nabla}\cdot\boldsymbol{v}^{n+1} = 0 \tag{2}$$

$$(\boldsymbol{v}^n\cdot\boldsymbol{\nabla})\boldsymbol{v}^{n+1} = -\boldsymbol{\nabla}p^{n+1} + \nabla^2\boldsymbol{v}^{n+1} \tag{3}$$

The implementation of the method begins with the distribution of the pressure and the velocity field at the nth iteration, which we denote by p^n and $\boldsymbol{v}^n$ respectively. The momentum equation can therefore be solved to give an estimation of the velocity field which is denoted by $\boldsymbol{v}^*$.

$$(\boldsymbol{v}^n\cdot\boldsymbol{\nabla})\boldsymbol{v}^* = -\boldsymbol{\nabla}p^n + \nabla^2\boldsymbol{v}^* \tag{4}$$

The estimated velocity field $\boldsymbol{v}^*$ does not in general satisfy the continuity equation. Hence, we introduce the velocity correction $\boldsymbol{\delta v}$ and the pressure correction δp, which are defined by the relations

$$\boldsymbol{v}^{n+1} = \boldsymbol{v}^* + \boldsymbol{\delta v}, \quad p^{n+1} = p^n + \delta p \tag{5}$$

Substituting the relations (5) in Eq. (2) we obtain the following equation for the velocity correction

$$\boldsymbol{\nabla}\cdot\boldsymbol{\delta v} = \mathrm{G} \quad \text{where} \quad \mathrm{G} = -\boldsymbol{\nabla}\cdot\boldsymbol{v}^* \tag{6}$$

where the term G depends only on $\boldsymbol{v}^*$ and it can be easily evaluated. At this point we introduce the scalar potential correction $\delta\varphi$ and the vector potential correction $\boldsymbol{\delta\psi}$, which are defined by the relation

$$\boldsymbol{\delta v} = \boldsymbol{\nabla}\delta\varphi + \boldsymbol{\nabla}\times\boldsymbol{\delta\psi} \tag{7}$$

It is noted that the velocity correction, as any vector, can be written as the sum of two terms $\boldsymbol{\delta v} = \boldsymbol{\delta v}_{\mathrm{ir}} + \boldsymbol{\delta v}_{\mathrm{sol}}$, one irrotational for which $\boldsymbol{\nabla}\times\boldsymbol{\delta v}_{\mathrm{ir}} = \boldsymbol{0}$ and hence $\boldsymbol{\delta v}_{\mathrm{ir}} = \boldsymbol{\nabla}\delta\varphi$ and one solenoidal for which $\boldsymbol{\nabla}\boldsymbol{\delta v}_{\mathrm{sol}} = 0$ and hence $\boldsymbol{\delta v}_{\mathrm{sol}} = \boldsymbol{\nabla}\times\boldsymbol{\delta\psi}$. Eq. (6) then becomes

$$\boldsymbol{\nabla}\cdot\boldsymbol{\delta v} = \boldsymbol{\nabla}\cdot(\boldsymbol{\delta v}_{\mathrm{ir}} + \boldsymbol{\delta v}_{\mathrm{sol}}) = \nabla^2\delta\varphi + \boldsymbol{\nabla}\cdot\boldsymbol{\nabla}\times\boldsymbol{\delta\psi} = \nabla^2\delta\varphi \tag{8}$$

Hence, combining Eq. (6) with Eq. (8) we obtain the auxiliary potential correction equation

$$\nabla^2\delta\varphi = \mathrm{G}. \tag{9}$$

Under this consideration, one may take the velocity correction that is necessary to satisfy continuity to be an irrotational vector $\boldsymbol{\delta v} = \boldsymbol{\delta v}_{\mathrm{ir}}$ without violation of the nature of the flow and of the actual velocity. This is due to the fact that the velocity correction is introduced in order to impose the mass conservation condition to the solution and not to correct the velocity in general. The solenoidal component of the velocity correction cannot be affected by the continuity equation (which expresses mass conservation) and thus it can be ignored. This leads to

$$\boldsymbol{\delta v} = \boldsymbol{\nabla}\delta\varphi \tag{10}$$

Thus, after solving the auxiliary potential correction Eq. (9) we compute the velocity correction from Eq. (10).

The boundary conditions for $\delta\varphi$ can be obtained in two ways. The exact velocity or velocity gradient is known on the boundaries so the velocity correction is zero. Hence:

(a) $\boldsymbol{\delta v}\cdot\hat{\mathbf{n}} = \boldsymbol{\nabla}\delta\varphi\cdot\hat{\mathbf{n}} = 0$ ($\hat{\mathbf{n}}$ is the unit normal vector on the boundary) gives the Neumann boundary condition $\partial\delta\varphi/\partial n = 0$.
(b) $\boldsymbol{\delta v}\cdot\hat{\mathbf{e}} = \boldsymbol{\nabla}\delta\varphi\cdot\hat{\mathbf{e}} = 0$ ($\hat{\mathbf{e}}$ is the unit tangential vector on the boundary) gives the condition $\partial\delta\varphi/\partial e = 0$. Supposing that at an arbitrary point of the boundary the potential correction $\delta\varphi$ assumes a value $\delta\varphi = c$ (which may be easily set to zero $c = 0$) it is obtained from $\partial\delta\varphi/\partial e = 0$ that $\delta\varphi = c$ (or $\delta\varphi = 0$) on the whole boundary. This gives a Dirichlet boundary condition.

Moreover, in the case of open boundaries, the conditions are subject to the global constraint

$$\int_c \frac{\partial\delta\varphi}{\partial n}ds = \int_s \delta^2\delta\varphi dA = \int_s GdA \tag{11}$$

where C is the boundary of the computational domain S.

Eq. (9) can be solved to give the potential correction $\delta\varphi$ using either one of the boundary conditions. However, numerical experiments with the present scheme showed that the Dirichlet boundary condition leads to faster convergence.

After the determination of the velocity correction field, the pressure correction needs to be computed. According to the present methodology, we must first express the convection terms of the momentum equation according to the vector identity