Journal of Computational Physics 365 (2018) 135–148

Contents lists available at ScienceDirect

Journal of Computational Physics

www.elsevier.com/locate/jcp

An oscillation-free flow solver based on flux reconstruction

Horacio J. Aguerre [a,e,*], Cesar I. Pairetti [b,a], Cesar M. Venier [a,b], Santiago Márquez Damián [a,c], Norberto M. Nigro [a,d]

[a] Centro de Investigación de Métodos Computacionales, CONICET-UNL, Santa Fe, Argentina
[b] Escuela de Ingeniería Mecánica, Facultad de Ciencias Exactas, Ingenieria y Agrimensura, Universidad Nacional de Rosario, Rosario, Argentina
[c] Facultad Regional Santa Fe, Universidad Tecnológica Nacional, Santa Fe, Argentina
[d] Facultad de Ingeniería y Ciencias Hídricas, Universidad Nacional del Litoral, Santa Fe, Argentina
[e] Facultad Regional Concepción del Uruguay, Universidad Tecnológica Nacional, Concepción del Uruguay, Argentina

ARTICLE INFO

Article history:
Received 12 January 2018
Accepted 21 March 2018
Available online 27 March 2018

Keywords:
Velocity oscillations
Flux reconstruction
Segregated method
Collocated grids
Finite Volume Method

ABSTRACT

In this paper, a segregated algorithm is proposed to suppress high-frequency oscillations in the velocity field for incompressible flows. In this context, a new velocity formula based on a reconstruction of face fluxes is defined eliminating high-frequency errors. In analogy to the Rhie–Chow interpolation, this approach is equivalent to including a flux-based pressure gradient with a velocity diffusion in the momentum equation. In order to guarantee second-order accuracy of the numerical solver, a set of conditions are defined for the reconstruction operator. To arrive at the final formulation, an outlook over the state of the art regarding velocity reconstruction procedures is presented comparing them through an error analysis. A new operator is then obtained by means of a flux difference minimization satisfying the required spatial accuracy. The accuracy of the new algorithm is analyzed by performing mesh convergence studies for unsteady Navier–Stokes problems with analytical solutions. The stabilization properties of the solver are then tested in a problem where spurious numerical oscillations arise for the velocity field. The results show a remarkable performance of the proposed technique eliminating high-frequency errors without losing accuracy.

1. Introduction

In the context of the Finite Volume Method (FVM), segregated algorithms are among the most popular strategies to couple pressure and velocity in Computational Fluid Dynamics (CFD). In these methodologies, the variables of the problem can be stored in the same or different spatial location, which gives place to grid arrangements known as collocated or staggered respectively. The staggered grids have the advantage of avoiding interpolations and thus prevent the appearance of spurious oscillations on the unknown fields. On the other hand, collocated grid arrangements are more suitable for unstructured meshes which are mandatory to address practical applications involving complex geometries.

One major disadvantage of collocated grids is the appearance of the aforementioned high-frequency oscillations due to a decoupling between pressure and velocity. This is related to the employment of central differencing for the discretization of the spatial derivatives. A common way to tackle these numerical issues is introducing a stabilizing effect on the pressure

* Corresponding author.
E-mail address: aguerrehoracio@gmail.com (H.J. Aguerre).

https://doi.org/10.1016/j.jcp.2018.03.033
0021-9991/

equation through the so-called Rhie–Chow interpolation [1]. Since its development, this technique has been the most used alternative to mitigate pressure oscillations and it is still subject of several discussions regarding its accuracy and stability properties [2–8]. One of the main issues regarding this technique is that it does not guarantee non-oscillatory fields for all problem conditions. Oxtoby et al. [6] and Mencinger et al. [7] proposed new discretization rules to eliminate unphysical spikes at the interface with porous media in Volume Of Fluid techniques. Other authors proposed solutions involving adjustments on the Rhie–Chow interpolation to mitigate these problems. In particular, Zhang et al. [8] put in evidence the flaws of the standard Rhie–Chow interpolation when the corrective terms on the mass conservation equations become sufficiently large. Such situation arises when dealing with discontinuous momentum sources. They propose a correction consisting on smoothing out the pressure gradient on cell faces by introducing the body forces in the calculation. In the same manner, Nordlund et al. [9] addressed similar problems and proposed two methods to mitigate spurious oscillations: one involving an adjustment of the porous resistivity in the cells near the interface and another one which introduces auxiliary pressure values in the same region. All these works propose particular solutions to eliminate oscillatory fields in different situations. However, for the velocity, a deeper study on the reasons that generate high-frequency oscillations still needs to be addressed, which is the main motivation of this work.

In this context, a new approach to mitigate spurious high-frequency velocity oscillations in pressure-velocity coupling algorithms for incompressible flows is presented; this formulation is inspired by the non-oscillatory nature of the fluxes in a collocated FVM. More precisely, the velocity field is defined by a face-to-cell reconstruction of the fluxes which must satisfy certain conditions to preserve second-order accuracy. Reconstruction operators can be obtained through different procedures, such as flux difference minimization [10–12] or weighted average [13]. Here, a reconstruction operator is designed based on a flux difference minimization to obtain an oscillation-free algorithm.

The paper is organized as follows: On Section 2, the general framework of the standard SIMPLE (Semi-Implicit Method for Pressure-Linked Equations) family algorithms is presented, including a study of the reasons for the velocity oscillations and a solution based on face-to-cell reconstruction. Section 3 presents an overlook of some reconstruction procedures followed by the development of a new reconstruction method. In Section 4, two benchmark cases are solved to demonstrate that the new formulation is second-order accurate. Section 5 shows the stabilization properties solving a porous media flow problem. Finally, the conclusions are presented in Section 6.

2. Theoretical background

2.1. Segregated algorithms on collocated grids

The continuum incompressible Navier–Stokes equations may be presented as:

$$\nabla\cdot \boldsymbol{u} = 0, \tag{2.1}$$

$$\frac{\partial \boldsymbol{u}}{\partial t} + \nabla\cdot(\boldsymbol{u}\boldsymbol{u}) = -\nabla p + \nu\,\nabla^2 \boldsymbol{u} + \boldsymbol{\Phi}, \tag{2.2}$$

where $p = \dfrac{\mathcal{P}}{\rho}$, $\mathcal{P}$ is the pressure field, $\boldsymbol{u} = (u, v, w)$ is the velocity field, ρ is the mass density, ν is the kinematic viscosity and $\boldsymbol{\Phi}$ is a momentum source term (e.g. a body force). Based on the Finite Volume Method [14], Eqs. (2.1) and (2.2) may be discretized as:

$$\sum_f \boldsymbol{u}_f\cdot \mathbf{S}_f = \sum_f F_f = 0, \tag{2.3}$$

$$a_P\,\boldsymbol{u}_P + \sum_N a_N\,\boldsymbol{u}_N = b_P\,\boldsymbol{u}_P^0 + \boldsymbol{\Phi}_P - \nabla p_P, \tag{2.4}$$

where the subscript P indicates the cell-centered value of the current cell, N indicates neighbor cells values and f refers to the face values. The term $b_P\,\boldsymbol{u}_P^0$ is the contribution of the previous time-step and is function of the temporal scheme. The face-normal vector $\mathbf{S}_f$ has the magnitude of the face area and points out of the cell, F_f is the velocity flux at the face f and a_P and a_N are the diagonal and off-diagonal momentum matrix coefficients respectively. The operator ∇p_P is a Gauss-based gradient of the pressure computed using first neighbors values of that field. Isolating $\boldsymbol{u}_P$ from Eq. (2.4),

$$\boldsymbol{u}_P = \boldsymbol{H}_P - \frac{1}{a_P}\nabla p_P, \tag{2.5}$$

where the term $\boldsymbol{H}_P$ is computed as,

$$\boldsymbol{H}_P = \frac{b_P\,\boldsymbol{u}_P^0 + \boldsymbol{\Phi}_P - \sum_N a_N\,\boldsymbol{u}_N}{a_P}. \tag{2.6}$$

One way to solve Eqs. (2.3) and (2.4) for $\boldsymbol{u}$ and p consists on addressing one equation at a time, update one of the unknown fields, solve the remaining equation and iterate the sequence. These techniques are known as segregated methods,