Computers & Fluids 88 (2013) 189–199

Contents lists available at ScienceDirect

Computers & Fluids

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

A meshless scheme for incompressible fluid flow using a velocity–pressure correction method

G.C. Bourantas [a], V.C. Loukopoulos [b,*]

[a] Applied Mathematics and Computational Science and Earth and Environmental Sciences and Engineering, King Abdullah University of Science and Technology, Thuwal, Saudi Arabia
[b] Department of Physics, University of Patras, Patras, 26500 Rion, Greece

ARTICLE INFO

Article history:
Received 15 April 2013
Received in revised form 15 August 2013
Accepted 9 September 2013
Available online 19 September 2013

Keywords:
Meshless method
Strong form
Incompressible flow
MLS
Velocity-pressure correction

ABSTRACT

A meshless point collocation method is proposed for the numerical solution of the steady state, incompressible Navier–Stokes (NS) equations in their primitive u–v–p formulation. The flow equations are solved in their strong form using either a collocated or a semi-staggered "grid" configuration. The developed numerical scheme approximates the unknown field functions using the Moving Least Squares approximation. A velocity, along with a pressure correction scheme is applied in the context of the meshless point collocation method. The proposed meshless point collocation (MPC) scheme has the following characteristics: (i) it is a truly meshless method, (ii) there is no need for pressure boundary conditions since no pressure constitutive equation is solved, (iii) it incorporates simplicity and accuracy, (iv) results can be obtained using collocated or semi-staggered "grids", (v) there is no need for the usage of a curvilinear system of coordinates and (vi) it can solve steady and unsteady flows. The lid-driven cavity flow problem, for Reynolds numbers up to 5000, has been considered, by using both staggered and collocated grid configurations. Following, the Backward-Facing Step (BFS) flow problem was considered for Reynolds numbers up to 800 using a staggered grid. As a final example, the case of a laminar flow in a two-dimensional tube with an obstacle was examined.

1. Introduction

Over the last years several numerical methods were proposed for the solution of the incompressible flow equations, namely, the continuity along with the momentum (Navier–Stokes) equations. The numerical solution of the Navier–Stokes non-linear Partial Differential Equations (PDEs) is a very challenging and demanding problem.

Concerning 2D flow problems, the governing equations are usually written in the conventional primitive variables (u–v–p) formulation [1]. Several methods have been developed for the numerical solution of the primitive type flow equations. In principle, the coupled velocity–pressure equations can be solved directly, as usually done in finite elements method and in some finite volume formulations [2]. On the other hand, very often, the transient problems are solved using an operator-splitting discretization scheme known as the projection method or the fractional step method [3–5]. This method developed in the 1960s, when Chorin observed that for incompressible flows the pressure was presented only as a Lagrange multiplier in order to enforce the incompressibility constraint. Due to the decoupling of the velocity and the pressure computations, the projection method is more efficient than the fully coupled procedures. For that reason, many improved projection methods have been seen in publications [6–11], utilizing traditional and well established numerical methods, such as Finite Element Method (FEM), the close related Finite Volume Method (FVM) and the Finite Difference Method (FDM).

Despite the extensive usage of projection methods, several methods were developed for the solution of the flow equations in their primitive variables formulation. Among them, the so-called penalty method has attracted quite an attention. Penalty method was initially introduced by Courant [12] in the context of the calculus of variations and its application to NS equations was initiated in [13]. It provides a convenient way to satisfy the incompressibility constraint by eliminating the pressure term. By this, it reduces the number of degrees of freedom and produces a more efficient and stable algorithm that requires only the calculation of the velocity field. Penalty method has numerous applications in several fields of science and engineering [14] [and references therein]. It can also be applied in the framework of the Finite Volume Method [15]. A widely used method is the Characteristic Based Split (CBS) method which is a fractional step method. The CBS scheme combines the characteristic Galerkin method with a splitting technique based on velocity correction. It can overcome the numerical instabilities and bypass the BB constraint since it uses equal order interpolations for both velocity and pressure variables. CBS scheme

* Corresponding author.
E-mail address: vxloukop@physics.upatras.gr (V.C. Loukopoulos).

0045-7930/$ - see front matter © 2013 Elsevier Ltd. All rights reserved.
http://dx.doi.org/10.1016/j.compfluid.2013.09.010

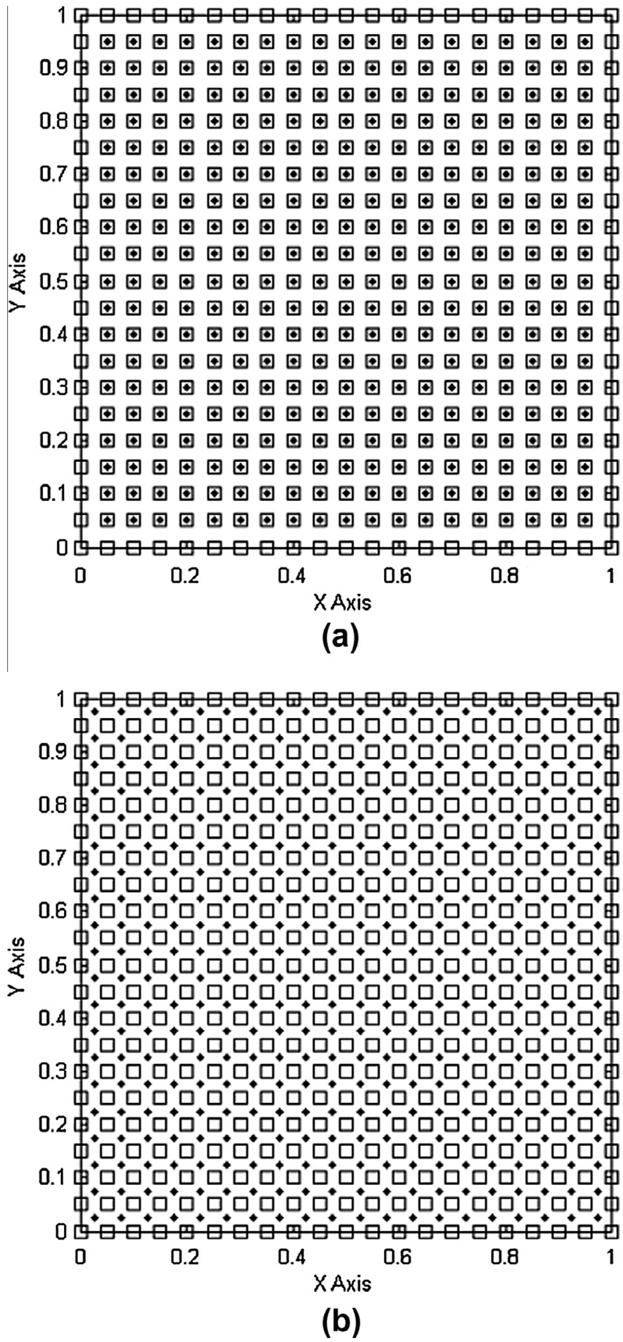

Fig. 1. (a) Collocated type uniform nodal distribution and (b) semi-staggered type uniform nodal distribution (square symbol-velocity nodes, dot symbol-pressure nodes).

has been successfully applied to many physical and engineering problems, such as compressible and incompressible flow [16,17], shallow water flow [18], free surface flow [19], turbulent flow [20], viscoelastic flow [21,22], porous medium flow [23] and solid dynamic [24].

Concerning the FVM, several methods were developed and applied for the numerical solution of a variety of physical and engineering flow problems. These are the methods that use a Poisson equation for the pressure, such as the SIMPLE (Semi-Implicit Method for Pressure Linked Equations) method [25] and its variants SIMPLER (Semi-Implicit Method for Pressure Linked Equations Revised) [26] and SIMPLEC (Semi-Implicit Method for Pressure Linked Equations Corrected) [27] that employ an iterative procedure for the velocity–pressure coupling. In details, the discretized continuity and momentum equations are combined in order to produce a discrete Poisson equation for the pressure. However, dealing with complex geometries can be a difficult task. Additionally, these methods require a staggered grid for the avoidance of spurious oscillations in the pressure. Another category of methods for incompressible flows in the context of $(u–v–p)$ formulation includes the SMAC method [28] that uses an auxiliary potential velocity for the determination of the pressure. Comparisons done with other numerical methods have shown that for unsteady flows the SMAC method is more efficient since less computational effort is needed [29].

Despite their applicability and the accuracy of the numerical results these methods had some serious drawbacks. Concerning the mesh based methods, FEM and FVM, the complex geometries used were hard to deal with. More precisely, the mesh generation is a difficult task especially for 3D simulations. The main effort during the simulation is spent on the stage of mesh generation. On the other hand, the grid based methods, such as FDM provide accurate results in regular geometries but it is difficult to deal with fully irregular geometries. Concluding, the main drawback is the refinement process especially at the regions of the spatial domain where the accuracy must be higher. For that reason, meshless methods are recently emerged as a promising alternative to overcome the problems of mesh generation and local refinement. The meshless method requires only a set of nodes, uniformly or randomly distributed along the interior and on the boundaries of a spatial domain, without prior knowledge for interconnectivity among the nodes. Numerous publications can be found regarding the numerical solution of the flow equations using meshless methods. These methods solve the flow equations either in their primitive variables formulation or in the velocity vorticity and streamfunction–vorticity formulation, using several meshless methods such as Meshless Local Petrov–Galerkin (MLPG) [30,31], Local Boundary Integral Equation (LBIE) [32,33], Meshless Point Collocation (MPC) [34,35], Element Free Galerkin (EFG) [36] and Diffuse Approximate Method (DAM) [37]. In the majority of the methods the calculation of pressure was done explicitly, giving the boundary conditions of the pressure or it was calculated as a final outcome, given the boundary conditions for pressure.

In a more detailed review of the meshless literature for fluid flow problems, in the context of DAM authors in [38–40] studied and analyzed the natural convection in fluids, in both 2D and 3D. Additionally, in [41] studied the natural convection Darcy inside a differentially heated cavity filled with a porous media, while the numerical solution of the Navier–Stokes equations for the lid-driven cavity flow problem for $Re = 5000$ was studied in [42]. The Radial Basis Function Collocation Method (RBFCM) has been extensively used for the numerical solution of partial differential equations and, recently started to be applied in many scientific and engineering disciplines related to fluid flow and transport. The method has been applied to natural convection flow problem by using asymmetric collocation in [43] and by symmetric modified collocation in [44]. The RBFCM has been, also, developed for diffusion problems [45], to convection–diffusion problems with phase-change [46], to industrial application of direct chill casting of aluminum alloys [47], to continuous casting of steel [48] and to solution of Navier Stokes equations [49]. Additionally, the method has been reformulated, in a way that the RBF's are derived by integrating the partial derivatives [50] and applied to transient problems [51] fluid flow [52] and moving boundaries [53].

The subject that is discussed herein concerns the numerical solution of the NS flow equations, in their primitive variables formulation, utilizing the meshless method and using a collocated or semi-staggered grid arrangement. The continuity equation and the Navier–Stokes equations are solved in the velocity–pressure