Journal of Computational Physics 277 (2014) 138–162

Contents lists available at ScienceDirect

Journal of Computational Physics

www.elsevier.com/locate/jcp

A multi-moment finite volume method for incompressible Navier–Stokes equations on unstructured grids: Volume-average/point-value formulation

Bin Xie [a], Satoshi Ii [b], Akio Ikebata [c], Feng Xiao [a,*]

[a] Department of Energy Sciences, Tokyo Institute of Technology, 4259 Nagatsuta Midori-ku, Yokohama, 226-8502, Japan
[b] Graduate School of Engineering Science, Osaka University, 1-3 Machikaneyama, Toyonaka, Osaka, 560-8531, Japan
[c] Production Technology Division, TOTO LTD, 1-1 Nakashima 2-chome, Kokurakita-ku, Kitakyushu, 802-8601, Japan

ARTICLE INFO

Article history:
Received 11 October 2013
Received in revised form 30 July 2014
Accepted 4 August 2014
Available online 12 August 2014

Keywords:
Finite volume method
Unstructured grid
Robustness
Accuracy
Triangular/tetrahedral mesh
Incompressible flow
Multi-moment
Compact-stencil

ABSTRACT

A robust and accurate finite volume method (FVM) is proposed for incompressible viscous fluid dynamics on triangular and tetrahedral unstructured grids. Differently from conventional FVM where the volume integrated average (VIA) value is the only computational variable, the present formulation treats both VIA and the point value (PV) as the computational variables which are updated separately at each time step. The VIA is computed from a finite volume scheme of flux form, and is thus numerically conservative. The PV is updated from the differential form of the governing equation that does not have to be conservative but can be solved in a very efficient way. Including PV as the additional variable enables us to make higher-order reconstructions over compact mesh stencil to improve the accuracy, and moreover, the resulting numerical model is more robust for unstructured grids.

We present the numerical formulations in both two and three dimensions on triangular and tetrahedral mesh elements. Numerical results of several benchmark tests are also presented to verify the proposed numerical method as an accurate and robust solver for incompressible flows on unstructured grids.

1. Introduction

The finite volume method (FVM) has gained a great popularity in computational fluid dynamics (CFD) because of its better conservativeness and flexibility to adapt to both structured and unstructured grids. Nearly all commercial CFD codes are based on the "FVM + unstructured grid" prevailing paradigm. However, developing high-order finite volume model on unstructured grids is not a trivial task. Conventional FVM requires wide stencil to generate high-order reconstructions as in the k-exact finite volume method [3] and the weighted essentially non-oscillatory method [13,19]. However, the choice of the cell stencil is not straightforward. As mentioned in [13], particular attention must be paid to choose the admissible stencils and there is not a simple rule to guide this procedure for reconstructions higher than first order. As a matter of fact, most operational codes, including the major commercial codes, are limited to the second order on unstructured grids,

* Corresponding author.
E-mail address: xiao@es.titech.ac.jp (F. Xiao).

http://dx.doi.org/10.1016/j.jcp.2014.08.011

where a linear reconstruction such as the MUSCL (Monotone Upstream-centered Schemes for Conservation Laws) scheme [38,39] is used.

The numerical schemes with compact stencil, on the other hand, add the degrees of freedoms (DOFs) locally on each cell for high order reconstructions, and are more flexible and easier to implement on unstructured meshes. Among many others, the representative methods of this sort are the discontinuous Galerkin (DG) method [7–10] and the spectral finite volume (SV) method [40,41]. These high order schemes were originally devised for convection-dominant flow problems, and make use of the Riemann solvers for the full Euler flux functions across cell boundaries where the reconstructed physical fields are allowed to be discontinuous. These schemes turned out to be a great success in solving convection-dominated flows. However, not as many as implementations of these methods are found for the incompressible Navier–Stokes equations where one has substantial difficulty because of the singularity in the hyperbolic system as the sound speed can be infinite.

The majority of numerical methods for incompressible flows are essentially based on the pressure-correction (or pressure-projection) approach, even they appear in the literature as different variants, such as the projection method [6], MAC (Marker and Cell) [17] method, SIMPLE (Semi-Implicit Method for Pressure-Linked Equations) method [29], SIMPLEC (SIMPLE Corrected) method [37] and PISO (Pressure Implicit with Splitting Operators) method [22]. All these methods work very well under the second-order FVM framework where the velocity can be directly coupled with the pressure through a pressure-projection step when staggering or co-volume arrangement is adopted. Although the second-order FVM on unstructured meshes proves itself a good trade-off between computational complexity and numerical accuracy, and has been widely accepted as a practical CFD framework for real-case applications, some remaining problems, for example the dependency of solution quality on computational mesh and the poor accuracy in advection computation, still deserve further efforts for more accurate and robust formulations. Some high-order DG methods have been devised for incompressible unsteady Navier–Stokes equations [32,18,12,34]. A common practice is to use the fractional-step pressure projection for the temporal updating and use DG for the spatial discretization, which is usually more complex and computationally expensive compared to the conventional FVM.

In this paper, we propose a novel numerical formulation to solve incompressible unsteady Navier–Stokes equations by including the point value (PV) of the velocity field at the cell vertices as a new DOF in addition to the volume integrated average (VIA) that is the only computational variable in the conventional FVM. The present method is called Volume integrated average and Point value based Multi-moment (VPM) method, in which the VIA moment is computed by a finite volume formulation of flux form, and thus exactly conservative, while the PV moment is point-wisely updated by a differential form that can be computed very efficiently in unstructured grids. The reconstruction that is needed to evaluate the numerical fluxes and the derivatives in the spatial discretization is determined from the constraint conditions in terms of both VIA and PV as the computational variables. More precisely, we construct piecewise incomplete quadratic polynomials of 6 DOFs in 2 and 8 DOFs in 3 dimensions. In addition to one VIA and three (2D) or four (3D) PVs of a single grid cell, the derivatives at the cell center are also used to determine the polynomials. These derivatives are approximated from the VIA and PVs of the cells sharing the boundary edges with the target cell. The stencil is compact, and more importantly there is no arbitrariness in choosing the stencil for reconstruction. As justified in this paper, without large increase in algorithmic complexity and computational cost, significant improvements are achieved in (1) the convergence rate in advection computation, (2) the accuracy of the whole fluid solver in terms of numerical errors and (3) the robustness to the computational mesh.

The present formulation can be seen as an extension of the CIP multi-moment finite volume methods [47,48,46,43,45,20,44,21,1,4] to incompressible Navier–Stokes equations on unstructured grids with triangular and tetrahedral elements. The multi-moment finite volume methods have been developed on structured grids as accurate and robust fluid solvers and applied so far to various applications. The underlying idea to make use of multiple types of moments facilitates novel numerical models of greater efficiency and flexibility and is consequently much beneficial for implementations in unstructured grids.

This paper is organized as follows. The numerical formulation is presented with details in Section 2. Numerical tests are given in Section 3 to verify the accuracy and the robustness of the present method in comparison with some standard FVM schemes. We end this paper by some conclusion remarks in Section 4.

2. Numerical formulation on unstructured mesh

We consider the incompressible unsteady Navier–Stokes equations,

$$\nabla \cdot \mathbf{u} = 0, \tag{1}$$

$$\frac{\partial \mathbf{u}}{\partial t} + \nabla \cdot (\mathbf{u} \otimes \mathbf{u}) = -\frac{1}{\rho}\nabla p + \nu \nabla^2 \mathbf{u}, \tag{2}$$

where $\mathbf{u} = (u, v, w)$ is the velocity vector with components u, v and w in x, y and z directions respectively. p is the pressure, ρ the density and ν the kinematic viscosity.