Journal of Computational Physics 312 (2016) 175–191

Contents lists available at ScienceDirect

Journal of Computational Physics

www.elsevier.com/locate/jcp

Discrete exterior calculus discretization of incompressible Navier–Stokes equations over surface simplicial meshes

Mamdouh S. Mohamed [a,*,1], Anil N. Hirani [b], Ravi Samtaney [a]

[a] *Mechanical Engineering, Physical Science and Engineering Division, King Abdullah University of Science and Technology, Jeddah, Saudi Arabia*
[b] *Department of Mathematics, University of Illinois at Urbana–Champaign, IL, USA*

A R T I C L E   I N F O

Article history:
Received 5 August 2015
Received in revised form 9 February 2016
Accepted 10 February 2016
Available online 11 February 2016

Keywords:
Discrete exterior calculus (DEC)
Navier–Stokes
Incompressible flow
Covolume method

A B S T R A C T

A conservative discretization of incompressible Navier–Stokes equations is developed based on discrete exterior calculus (DEC). A distinguishing feature of our method is the use of an algebraic discretization of the interior product operator and a combinatorial discretization of the wedge product. The governing equations are first rewritten using the exterior calculus notation, replacing vector calculus differential operators by the exterior derivative, Hodge star and wedge product operators. The discretization is then carried out by substituting with the corresponding discrete operators based on the DEC framework. Numerical experiments for flows over surfaces reveal a second order accuracy for the developed scheme when using structured-triangular meshes, and first order accuracy for otherwise unstructured meshes. By construction, the method is conservative in that both mass and vorticity are conserved up to machine precision. The relative error in kinetic energy for inviscid flow test cases converges in a second order fashion with both the mesh size and the time step.

1. Introduction

When solving a partial differential equation numerically, various measures (e.g. convergence, stability and consistency) are usually investigated to verify the implemented discretization. Such measures, although reflecting the mathematical fidelity of the discretization, may not give insight into the physical fidelity of the discretization. By physical fidelity we mean how well does the discrete system of equations conserve secondary quantities, such as kinetic energy, implied in the continuous equation but not explicitly constructed or built into the numerical scheme. The development of such physically conservative discretizations, for Navier–Stokes (NS) equations for example, is favorable for many physical applications (e.g. turbulent flows and shallow-water simulations) to avoid undesirable numerical artifacts. Among other discretization approaches, some staggered mesh schemes are known for their conservation of both primary (i.e. mass and momentum) and secondary (e.g. kinetic energy and vorticity) physical quantities [1].

Staggered mesh methods were first developed by Harlow and Welch [2] for structured Cartesian meshes by placing the velocity and pressure degrees of freedom at different positions on the mesh. Later on, the approach was extended to unstructured meshes by Nicolaides [3] and Hall et al. [4], which is now known as the covolume (or dual-variable) method.

* Corresponding author.
E-mail addresses: mamdouh.mohamed@kaust.edu.sa (M.S. Mohamed), hirani@illinois.edu (A.N. Hirani), ravi.samtaney@kaust.edu.sa (R. Samtaney).
[1] On leave, Department of Mechanical Design and Production, Faculty of Engineering, Cairo University, Giza, Egypt.

http://dx.doi.org/10.1016/j.jcp.2016.02.028

The covolume method was originally introduced as a low order method that does not experience spurious modes that were common in low order discretizations of viscous flows. The derivation of such discretization commences by taking the dot product of the momentum equation with the unit normal vector to each face of the triangular/tetrahedral elements. This reduces the velocity vector field to a scalar flux (equal to the mass flux across the face for an incompressible flow with constant density) defined on each face. In this approach, the pressure is consequently defined at the circumcenter of each triangular/tetrahedral element. The accuracy of the covolume scheme was estimated by Nicolaides [5] to be second order for a mesh with all its triangular elements having the same circumradii (i.e. structured-triangular mesh) and first order accurate otherwise. These accuracy estimates were in agreement with numerical experiments [4]. Several forms of the covolume method were then developed for both two-dimensional (2D) (only on planar domains) and three-dimensional (3D) domains, where the difference was mainly in the convective term discretization [6–9].

The conservation properties of the covolume method were later revealed by Perot [10]. The divergence form of Navier–Stokes equations was proved to conserve the momentum and kinetic energy both locally and globally. On the other hand, the rotational form of Navier–Stokes equations was found to conserve the circulation and kinetic energy locally and globally for both 2D [10] and 3D [11] discretizations. These conservation properties of the covolume method, in addition to the attractive properties of its differential operators that mimic the behavior of their continuous counterparts, shed light on the merit of using discrete calculus methods to solve other physics problems [12].

Another approach to develop conservative discretizations for incompressible flows emerged from the computer graphics community, aiming to mitigate the effects of numerical viscosity that causes detrimental visual consequences [13,14]. In this approach, the Navier–Stokes equations were discretized through the discrete exterior calculus (DEC) framework; the discretization of the smooth exterior calculus operators [15,16]. A main advantage of DEC discretizations is the applicability to simulate flows over curved surfaces, unlike the covolume approach. The resulting discrete equations had similarities with the covolume method, with the differences mainly in the convective term discretization. In practice, the convective term was not discretized using DEC but employed a method of characteristics with an interpolation scheme based on Kelvin's circulation theorem [13], or a finite volume based approach [14]. However, the presented numerical test cases using the DEC approach lacked comprehensive quantitative analysis of the scheme's accuracy and its conservative behavior.

This paper presents a discretization of the Navier–Stokes equations through discrete exterior calculus. Hence, similar to previous DEC-based discretizations, the developed discretization is capable of simulating flows over curved surfaces, which distinguishes it from the covolume method. In addition, the convective term in the presented discretization is different from previous DEC-based and covolume discretizations. The Navier–Stokes equations are first rephrased using the exterior calculus notation in Section 2. The DEC discretization of NS equations is then derived in Section 3 for both 2D and 3D cases, highlighting its distinction from the covolume method and previous DEC-based discretizations. In Section 4, several numerical experiments are illustrated for incompressible flows over 2D flat/curved domains to benchmark the convergence and conservative behavior of the developed scheme. The paper closes with conclusions summarizing the main features of the presented discretization, and addressing potential future developments.

2. Navier–Stokes equations in exterior calculus notation

The first step in deriving a DEC discretization of NS equations is to express the equations using the exterior calculus notation. This is done first by starting from the well-known vector calculus formulation of NS equations (in Euclidean space) and substituting with identities relating the differential operators; viz. div, grad and curl, with exterior calculus operators; viz. exterior derivative, Hodge star and wedge product. An alternative derivation of the resultant formulation is then presented, starting from the coordinate invariant formulation of NS equations expressed in terms of the Lie and exterior derivatives. Readers not familiar with exterior calculus may refer to [17,18] for a concise introduction to the topic.

Considering the incompressible flow of a homogeneous fluid with unit density and no body forces, the governing equations for the fluid motion are the Navier–Stokes equations expressed as

$$\frac{\partial \mathbf{u}}{\partial t} - \mu \Delta \mathbf{u} + (\mathbf{u}.\nabla)\mathbf{u} + \nabla p = 0, \tag{1a}$$

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

where $\mathbf{u}$ is the velocity vector, p is the pressure and μ is the dynamic viscosity. Substituting with the tensor identities: $\Delta\mathbf{u} = \nabla(\nabla.\mathbf{u}) - \nabla \times (\nabla \times \mathbf{u})$ and $(\mathbf{u}.\nabla)\mathbf{u} = \frac{1}{2}\nabla(\mathbf{u}.\mathbf{u}) - \mathbf{u} \times (\nabla \times \mathbf{u})$, and considering the incompressibility condition $\nabla.\mathbf{u} = 0$, Eq. (1a) can be expressed in its rotational form as

$$\frac{\partial \mathbf{u}}{\partial t} + \mu \nabla \times \nabla \times \mathbf{u} - \mathbf{u} \times (\nabla \times \mathbf{u}) + \nabla p^d = 0, \tag{2}$$

where p^d is the dynamic pressure defined as $p^d = p + \frac{1}{2}(\mathbf{u}.\mathbf{u})$.

The notation transformation is carried out by applying the flat operator ($\flat$) to Eqs. (2) and (1b), and substituting with the following identities