Journal of Computational Physics 321 (2016) 76–104

Contents lists available at ScienceDirect

Journal of Computational Physics

www.elsevier.com/locate/jcp

A consistent direct discretization scheme on Cartesian grids for convective and conjugate heat transfer

Norikazu Sato [a,*], Shintaro Takeuchi [b], Takeo Kajishima [b], Masahide Inagaki [a], Nariaki Horinouchi [a]

[a] *Toyota Central R&D Labs., Inc., 41-1 Yokomichi, Nagakute, Aichi 480-1192, Japan*
[b] *Osaka University, 2-1 Yamada-oka, Suita, Osaka 565-0871, Japan*

ARTICLE INFO

Article history:
Received 18 October 2015
Received in revised form 25 February 2016
Accepted 18 May 2016
Available online 25 May 2016

Keywords:
Cartesian grid method
Fluid flow
Convective heat transfer
Conjugate heat transfer
Immersed boundary

ABSTRACT

A new discretization scheme on Cartesian grids, namely, a "consistent direct discretization scheme", is proposed for solving incompressible flows with convective and conjugate heat transfer around a solid object. The Navier–Stokes and the pressure Poisson equations are discretized directly even in the immediate vicinity of a solid boundary with the aid of the consistency between the face-velocity and the pressure gradient. From verifications in fundamental flow problems, the present method is found to significantly improve the accuracy of the velocity and the wall shear stress. It is also confirmed that the numerical results are less sensitive to the Courant number owing to the consistency between the velocity and pressure fields. The concept of the consistent direct discretization scheme is also explored for the thermal field; the energy equations for the fluid and solid phases are discretized directly while satisfying the thermal relations that should be valid at their interface. It takes different forms depending on the thermal boundary conditions: Dirichlet (isothermal) and Neumann (adiabatic/iso-heat-flux) boundary conditions for convective heat transfer and a fluid–solid thermal interaction for conjugate heat transfer. The validity of these discretizations is assessed by comparing the simulated results with analytical solutions for the respective thermal boundary conditions, and it is confirmed that the present schemes also show high accuracy for the thermal field. A significant improvement for the conjugate heat transfer problems is that the second-order spatial accuracy and numerical stability are maintained even under severe conditions of near-practical physical properties for the fluid and solid phases.

1. Introduction

Heat transfer and fluid flow problems are widely observed in industrial applications, and they are strongly related to the performance, efficiency, and reliability of these systems. Therefore, it is important to understand complicated flow and thermal phenomena and adequately control them. Computational fluid dynamics (CFD) has been extensively used at various stages of engineering, including feasibility studies, testing and improvement of prototypes, and manufacturing design processes, in order to reduce the time and cost of product development. In those simulations, boundary conforming structured and unstructured grids are commonly used. Although structured grids are suitable for simulations that require high grid

* Corresponding author.
E-mail address: nsato@mosk.tytlabs.co.jp (N. Sato).

http://dx.doi.org/10.1016/j.jcp.2016.05.034

resolutions to resolve thin boundary layers or unstable shear layers, they are not practical for complex geometries. Unstructured grids, such as tetrahedral grids, are comparatively appropriate for complex geometries, however, computational accuracy is strongly influenced by the quality of the mesh. Moreover, even with the use of unstructured grids, it is difficult to generate a computational mesh for extremely complex geometries, such as an automobile engine compartment. Thus, developing a simpler way to generate a mesh and shortening the duration of the total analysis cycle are crucial for using CFD in practical applications.

The Cartesian grid method, in which the underlying mesh does not need to conform with body geometries, is expected to be a feasible approach to overcome those issues. In the Cartesian grid method, there are a lot of variations for approximating the geometry. The voxel method [1], where the geometry is represented in a stepwise pattern, is a low-order approximation method, while higher-order methods include the immersed boundary (IB) method [2–7], the cut-cell method [8], the ghost fluid method [9,10], and the immersed interface method [11–13]. In general, application targets of the Cartesian grid method can be categorized into (a) flows with moving boundaries, (b) flows with complex geometries, and (c) multiphase flows. These methods are selected depending on desired accuracy and cost for the targets.

For incompressible flows, the immersed boundary method is probably the most widely employed Cartesian grid method. It is originated from the study by Peskin [2], where the presence of the solid body immersed in a fluid is modeled by the body-force on a fixed-rectangular grid by using the Dirac delta function. Because of its simple algorithm, the method and its evolved versions have been applied to various types of fluid–structure interaction problems of category (a) [2,14–16]. In order to relax the limitations on its numerical stability, especially for the rigid-body case, Mohd-Yusof [3] proposed the direct forcing approach, and further modifications suited to three-dimensional problems, were adopted by Fadlun et al. [4]. In the direct forcing approach, the velocity in the cell adjacent to an immersed boundary is obtained by interpolation between the velocities at the boundary and in the surrounding fluid cells. The method has found a variety of practical applications, primarily in category (b), e.g., flows inside an internal combustion piston assembly [4], in an impeller stirred tank [17], around a road vehicle model [18,19], and past a nuclear rod bundle [20]. Meanwhile, Ikeno and Kajishima [20] discussed the inconsistency between the discretized forms of the momentum and pressure equations in the direct forcing approaches. They proposed a consistent scheme in accordance with its forcing strategy and confirmed the effectiveness of this scheme for several fundamental flow problems. Although the mass conservation and the corresponding accuracy are drastically improved by this simple treatment of the velocity-pressure consistency, the numerical accuracy is not always improved, and may even deteriorated under the conditions of low grid resolutions. As will be shown below, a deterioration in accuracy is usually caused by a nature of the most Cartesian grid methods as the momentum conservation is not directly satisfied near the boundary, where some interpolation/extrapolation procedures are employed for satisfying the no-slip condition. Therefore, it should be emphasized that solving the governing equations even in the immediate vicinity of the boundary is as important as maintaining the consistency between the velocity and pressure fields.

The immersed interface method [11–13] is one of the fixed-grid methods achieving higher-order spatial accuracy. The method resolves discontinuous interface by incorporating the jump condition into a finite difference formulation, thereby maintaining second-order accuracies for velocity and even for pressure fields when it is coupled with a projection method for the incompressible Navier–Stokes equation [11]. In the meantime, the discretization scheme tends to be complicated especially for some situations where different physical properties are applied for contiguous two phases, e.g., a fluid–solid thermal interaction problem which is one of the targets of the present study.

The Cartesian grid method has been also applied to heat transfer problems in a fluid including/bounded by solid surfaces. Generally, those problems are described with heat convection in the fluid and conduction in both phases. However, in heat transfer simulations, for simpler treatment than in the conjugate problems, isothermal conditions (Dirichlet-type boundary conditions) as well as adiabatic or iso-heat-flux conditions (Neumann-type boundary conditions) are often used. Previous studies have mostly dealt with Dirichlet boundary conditions [21–24], and there are a limited number of studies that implement Neumann boundary conditions. Pacheco et al. [25] and Pacheco-Vega et al. [26] proposed a successive determination algorithm of the temperature inside a body to match the prescribed heat flux at the immersed boundary. Zhang et al. [27] proposed another implementation of a Neumann boundary condition; the temperature gradient at the boundary is evaluated on layers of Lagrangian points along the body surface. The validity of these algorithms has been ascertained for various problems, such as natural convection in an inclined cavity, heat diffusion in an annulus, and convective heat transfer around a stationary/oscillatory cylinder. However, these algorithms are complicated, since they require special internal iterations within a time step or arrangements of additional virtual points. Moreover, it encounters in natural and practical situations that local heat flux develops within the solid object where neither boundary condition is applicable at the interface.

In a conjugate heat transfer problem where fluid and solid thermal fields are coupled at their interface, certain thermal relations at the interface regarding the continuity of the temperature and heat flux have hampered the application of the Cartesian grid methods. Although solving the energy equation for the averaged temperature with a weight of volume of the fluid/solid fraction [28] is an easy way to perform this type of simulation, the thermal relation at the fluid–solid interface is not satisfied. Pioneering work was performed by Yu et al. [29], who extended the fictitious-domain method by incorporating the interfacial thermal relation and then applied it to a conjugate heat transfer problem. More recently, Shao et al. [30] combined the sharp-interface method with their fictitious-domain method in order to enhance its accuracy and computational efficiency. Kang et al. [31] proposed an implementation in their immersed boundary framework for a multi-material thermal interaction problem. Nagendra et al. [32] developed an IB method applicable to a boundary non-conforming mesh with curvilinear coordinates, in which treatments for various thermal boundary conditions for convective and conjugate