International Journal of Heat and Mass Transfer 104 (2017) 98–111

Contents lists available at ScienceDirect

International Journal of Heat and Mass Transfer

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

A ghost-cell immersed boundary method for the simulations of heat transfer in compressible flows under different boundary conditions Part-II: Complex geometries

Kun Luo [a], Chaoli Mao [a], Zhenya Zhuang [a], Jianren Fan [a,*], Nils Erland L. Haugen [b]

[a] State Key Laboratory of Clean Energy Utilization, Zhejiang University, Hangzhou 310027, PR China
[b] SINTEF Energy Research, N-7465 Trondheim, Norway

ARTICLE INFO

Article history:
Received 12 May 2016
Received in revised form 6 July 2016
Accepted 8 August 2016

Keywords:
Immersed boundary method
Compressible flow
Irregular boundary
Heat transfer

ABSTRACT

In this paper, our previous ghost-cell compressible immersed boundary method (Luo et al., 2016) is further implemented to solve heat transfer problems in flows with complex solid geometries. Arbitrary 2D-immersed boundaries are presented by many micro line segments. Each line segment is identified by two vertices. An extension to 3D situation is straightforward, in which arbitrary surfaces can be divided into many triangular surface elements. Two different interpolation schemes for the mirror points, namely inverse distance weighting and bilinear interpolations, are compared. An accurate capture of the secondary vortex street far behind an elliptical cylinder indicates a successful combination of current IB method with the fluid solver. Then, forced convective flow over an inclined non-circle cylinder is used to further validate present method. Finally, *Mach* > 0.3 cases are studied to demonstrate the essentiality of taking compressibility into consideration in high-speed thermal flow problems.

1. Introduction

The CFD (computational fluid dynamics) method has been a well-developed academic discipline and gradually become an effective instrument for engineering problems. When it comes to flows over complex geometries, in a traditional way, a body-fitted mesh is generated to describe the boundary of the immersed body. Under this situation, the implementation of boundary conditions is simple and straightforward because grid line and body surface align with each other. However, for arbitrary complex geometries, the generation process of a high quality body-conformal grid and its re-meshing process can be very resource-consuming.

In contrast, a totally different idea had been introduced by Peskin [2] in 1972. That is "immersed boundary method (IB method)", in which a Cartesian grid was used to resolve blood flow regardless of the complex geometry of human heart valve. The effect of elastic heart valve wall on surrounding fluid flow was taken into consideration through a force term on the right-hand side of momentum equation. This idea successfully avoids the generation of an unstructured body-fitted grid to conform complex geometries and thereby makes simulation of complex structure–fluid interaction more efficient. Moreover, as a result of using mesh of simple topology structure, the parallelization of code is straightforward and also more efficient.

Since then, IB methods have attracted many researchers' attention. Many efforts have been made to improve the accuracy and broaden the application. Generally, IB methods fall into two different categories, i.e., continuous force approach and discrete force approach [3]. A detailed discussion of IB methods can be found in [4–7]. IB method was originated to mimic the effect of elastic boundary on fluid flow and it made a sense that a force term was to represent the effect since the elastic force model in [1] had a physic basis. In a similar way, a PID (portion-integral-derivation) force model was presented for rigid boundary [8]. However, the free parameters included in this model may degrade numerical accuracy as well as stability. To overcome this downside, Fadlun et al. [9] proposed a direct-forcing scheme for rigid immersed body. They also showed that solving the interpolation formulas together with discrete momentum equations was equivalent to applying force term and then the explicit addition of force term was not required. This is where the original idea of ghost-cell based immersed boundary (GCIB) method comes from. Compared with original IB method [1] and direct-forcing method [10,11], no Dirac delta function is used to distribute the force term from Lagrange point to underlying Euler grid in GCIB method. Therefore, the boundary is sharply represented. This is a desirable feature to

* Corresponding author.
E-mail address: fanjr@zju.edu.cn (J. Fan).

http://dx.doi.org/10.1016/j.ijheatmasstransfer.2016.08.010
0017-9310/

resolve boundary layer in high Reynolds number flows. Besides, since in GCIB method there is no need to modify the fluid solver (i.e., the implementation of boundary conditions through this method can be summarized into a separate module), its combination with existing solver is very easy. Another point worth of noting is that a high-order interpolation scheme can be constructed for GCIB method [12,13] to further save computational resource.

The difficulty in GCIB method's extension to situations with irregular geometries lies in how to track the boundaries correctly. Conceptually, two ideas exist to overcome this. An unstructured triangle surface mesh was used in [14–16]. This mesh can be used to represent arbitrary geometries and has gained its popularity in biology fluid mechanics where a very complex body, such as a bluegill sunfish pectoral fin and a false vocal fold, interacts with surrounding flows in a two-way coupled manner [17,18]. Another choice is a standard level-set signed distance function [19,20]. This strategy also applies to both rigid and deformable structures. We refer the reader to [14–20] and references therein for detailed descriptions of these two methods.

Application of IB methods to heat transfer problems was reported by Kim et al. [21], in which a heat source/sink was introduced into energy conservation equation to account for the effect of hot/cool boundary wall. Wang et al. [22] proposed a multi-direct heat source scheme to improve the accuracy of boundary condition enforcement. Since then, many researchers have made their efforts to improve IB methods' capability to various heat transfer problems [23–29]. In our previous work [1], a second-order accurate GCIB method was designed for the implementation of Dirichlet, Neumann and Robin boundary conditions. It also should be noted that our GCIB method was combined with a compressible fluid solver [30] and study on the effect of compressibility on heat transfer process was carried out. In this paper, we combine our previous method [1] with an unstructured surface mesh to devise a GCIB method for the simulation of heat transfer process between compressible fluid and irregular boundaries. To our best knowledge, no such report has ever been presented.

The rest part of the current paper is organized as follows. Section 2 gives the numerical methodology, including compressible governing equations, introduction to GCIB method, construction strategy for irregular geometry and two different interpolation procedures for mirror point. Section 3 starts with a test case where effect of the relative resolution between boundary and background grid is investigated. Following this is a spatial convergence examination to check if the present GCIB method still remains a second-order accuracy. After these, several benchmark cases are studied to validate our GCIB method's capability to handle irregular fluid–solid interface. Furthermore, compressibility effect in high speed forced convective flow is revealed. Finally, we draw a conclusion in Section 4.

2. Numerical methodology

2.1. Governing equations

Mass, momentum and energy conservation equations together with the equation of state are used to describe the compressible flows in present paper. The continuity equation reads as follows,

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \vec{u}) = 0 \tag{1}$$

where ρ is fluid density, $\vec{u}$ is the vector of fluid velocity and t is time.

In our research, the temperature ratio between solid and fluid is small. Therefore, we can assume properties of fluid such as dynamic viscosity μ, specific heat c_p at constant pressure and heat conductivity λ to be constant. Thus, the momentum and energy conservation equations can be simplified as,

$$\frac{\partial \vec{u}}{\partial t} + \vec{u} \cdot \nabla(\vec{u}) = \frac{1}{\rho}\left(-\nabla p + \vec{f}_{vs}\right) \tag{2}$$

$$\frac{\partial T}{\partial t} + \vec{u} \cdot \nabla T = \frac{1}{\rho c_p}(\nabla \cdot (\lambda \nabla T) + 2\rho \nu S \otimes S) \tag{3}$$

In Eq. (2), p is pressure, $\vec{f}_{vs} = \nabla \cdot (2\rho\nu S)$ is viscous force, and $\nu = \mu/\rho$ is kinematic viscosity. The symbol S represents trace-less strain rate tensor with $S_{ij} = (\partial u_i/\partial x_j + \partial u_j/\partial x_i)/2 - \delta_{ij}\nabla \cdot \vec{u}/3$. The last term on the right hand side of energy equation is viscous dissipation source.

The state equation for an ideal gas to close Eqs. (1)–(3), is given by,

$$p = \rho R T \tag{4}$$

where, R is the specific gas constant and can be calculated from $R = R_u/M$. R_u is universal gas constant and M is molar mass.

The above governing equations are solved by a sixth-order centered finite-difference scheme on a non-staged mesh and an explicit three-stage Runge–Kutta scheme for spatial derivative and time advancement, respectively. The time step is limited by CFL number criterion. The sixth-order centered finite-difference scheme can be expressed as follows,

The first-order derivative:

$$f'_i = (-f_{i-3} + 9f_{i-2} - 45f_{i-1} + 45f_{i+1} - 9f_{i+2} + f_{i+3})/60\delta x \tag{5}$$

The second-order derivative:

$$f''_i = (2f_{i-3} - 27f_{i-2} + 270f_{i-1} - 490f_i + 270f_{i+1} - 27f_{i+2} + 2f_{i+3})/180\delta x^2 \tag{6}$$

where, δx is the local grid size.

To avoid "wiggles", the advection term in Eqs. (1)–(3) is discretized by a fifth-order upwinding scheme in which the point furthest downwind is excluded from the centered finite-difference stencil. The fifth-order upwinding stencil can be written as,

$$-uf'_{(up,5th)} = -uf'_{(centr,6th)} + \frac{|u|\Delta x^5}{60} f^{(6)} \tag{7}$$

where Δx is local grid size.

In order to construct the above upwinding scheme, the sixth-order derivative is needed. And it is straightforward to approximate such a derivative by Taylor expansion on a uniform mesh. However, when a stretched grid is used, things become complicated. In this paper, the following chain procedure is proposed to calculate the sixth-order derivative and thus to construct a fifth-order upwinding scheme on a stretched grid.

$$f'(x) = f'(\zeta)/x' \tag{8}$$

$$f''(x) = f''(\zeta)/(x')^2 - x''f'(\zeta)/(x')^2 \tag{9}$$

$$f'''(x) = f'''(\zeta)/(x')^3 - 3x''f''(\zeta)/(x')^2 - x'''f'(\zeta)/(x')^3 \tag{10}$$

$$f^{(4)}(x) = \frac{f^{(4)}(\zeta)}{(x')^4} - \frac{6x''}{(x')^2}f'''(\zeta) - \left[\frac{4x'''}{(x')^3} + \frac{3(x'')^2}{(x')^4}\right]f''(\zeta) - \frac{x^{(4)}}{(x')^4}f'(\zeta) \tag{11}$$

$$f^{(5)}(x) = \frac{f^{(5)}(\zeta)}{(x')^5} - \frac{10x''}{(x')^2}f^{(4)}(\zeta) - \left[\frac{10x'''}{(x')^3} + \frac{15(x'')^2}{(x')^4}\right]f'''(\zeta) - \left[\frac{5x^{(4)}}{(x')^4} + \frac{10x''x'''}{(x')^5}\right]f''(\zeta) - \frac{x^{(5)}}{(x')^5}f'(\zeta) \tag{12}$$