Applied Thermal Engineering 72 (2014) 29–33

Contents lists available at ScienceDirect

Applied Thermal Engineering

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

Simulations of a falling sphere with concentration in an infinite long pipe using a new moving mesh system

Shinichi Asao [a,*], Kenichi Matsuno [b], Masashi Yamakawa [b]

[a] College of Industrial Technology, Japan
[b] Kyoto Institute of Technology, Japan

HIGHLIGHTS

- New Trans-mesh method and Moving computational domain method have been introduced.
- In the Trans-mesh method, bodies can move freely with adding and removing meshes.
- The feature of the MCD method is to move computational domain with the body.
- We can simulate a falling sphere with concentration in an infinite long bending pipe.

ARTICLE INFO

Article history:
Received 29 September 2013
Accepted 27 June 2014
Available online 4 July 2014

Keywords:
Moving boundary problem
Incompressible flows
Finite volume method
Unsteady flow

ABSTRACT

The purpose of this paper is to simulate a flow around a falling sphere with concentration in an infinite long pipe using a new moving mesh system. New moving mesh system is called a Trans-mesh method and a Moving Computational Domain method. In the Trans-mesh method, the bodies can move freely in a main mesh that covers the entire flow field. On the other hand, in the Moving Computational Domain method, the whole of the computational domain including bodies inside moves in the physical space without the limit of region size. These methods are constructed based on the four-dimensional control volume in space-time unified domain such that the method satisfies both the physical and geometrical conservation laws simultaneously. As a result of simulations, physically-meaningful flows are obtained, and it is confirmed that the new moving mesh system is useful for this simulation.

1. Introduction

Today, one of the interesting problems in Computational Fluid Dynamics is an unsteady flow and it is very important to calculate a moving boundary problem. Especially, in the case that body moves in the fluid is interesting on engineering. When we simulate such flow field, we might encounter some problems to be overcome. One of the problems is on mesh system. When the body moves in the flow field, a conventional single body-fitted grid system is hard to adjust the motion of body. A shape of the computational mesh is highly skewed in the case that the body travels long distance in the flow field. To overcome this problem, we have proposed a Trans-mesh method [1] for three-dimensional space. The method is that the front mesh plane of the moving body is eliminated from the main mesh to avoid mesh folding according to decrease of the mesh spacing due to the movement of the body, while a mesh plane is added newly between the rear plane of the moving body and the main mesh in order to keep the allowable maximum mesh spacing. Next, we consider a flow around a moving sphere in the long pipe. It is necessary to make the computational mesh for whole of the pipe. Then, a huge number of computational mesh is needed. As the result, this simulation by using traditional method spends a lot of time. We have proposed the Moving Computational Domain method [2]. This method is kind of moving mesh method. The feature of this method is to move computational domain with the body. The Moving Computational Domain method can consider the region without limit. The only necessary assumption is that the conditions just in front of the computational domain should be known a priori, such as, stationary fluid state or uniform flow and so on. As these flow solvers, we modified the Moving-grid Finite-volume method [3]. The method is constructed based on the four-dimensional control volume in space-time (x,y,z,t) unified domain such that the method satisfies the divergence-free character in the (x,y,z,t) space and both the physical and geometrical conservation laws simultaneously [4]. Due to the use of four-dimensional control

* Corresponding author. Tel.: +81 6 6431 7354; fax: +81 6 6431 5998.
E-mail address: asao@cit.sangitan.ac.jp (S. Asao).

http://dx.doi.org/10.1016/j.applthermaleng.2014.06.059
1359-4311/

volume, the method has a lot of merits or freedom. The purpose of this paper is to extend the method to the system including advection-diffusion equation of concentration and to develop the Trans-mesh method and Moving Computational Domain method. The methods are applied to a falling sphere with concentration by gravity in an infinite long pipe.

2. New moving mesh system

2.1. Governing equations

Governing equations are the continuity equation, the incompressible Navier–Stokes equations and the advection-diffusion equation of concentration. These are written as follows:

$$\nabla \cdot \mathrm{q} = 0, \tag{1}$$

$$\frac{\partial q}{\partial t} + \frac{\partial E_a}{\partial x} + \frac{\partial F_a}{\partial y} + \frac{\partial G_a}{\partial z} = -\left(\frac{\partial E_p}{\partial x} + \frac{\partial F_p}{\partial y} + \frac{\partial G_p}{\partial z}\right) + \left(\frac{\partial E_v}{\partial x} + \frac{\partial F_v}{\partial y} + \frac{\partial G_v}{\partial z}\right), \tag{2}$$

$$\frac{\partial c}{\partial t} + \frac{\partial E_c}{\partial x} + \frac{\partial F_c}{\partial y} + \frac{\partial G_c}{\partial z} = \frac{\partial E_d}{\partial x} + \frac{\partial F_d}{\partial y} + \frac{\partial G_d}{\partial z} \tag{3}$$

where, q is the velocity vector, E_a, F_a, and G_a are advection flux vectors in the x, y, and z directions, respectively, E_v, F_v, and G_v are viscous flux vectors, and E_p, F_p, and G_p are pressure flux vectors, E_c, F_c, and G_c are convection flux, and E_d, F_d, and G_d diffusion flux. The elements of the velocity vector and flux vectors are:

$$\begin{aligned}
&\mathrm{q} = \begin{bmatrix} u \\ v \\ w \end{bmatrix}, \quad \mathrm{E}_a = \begin{bmatrix} u^2 \\ uv \\ uw \end{bmatrix}, \quad \mathrm{F}_a = \begin{bmatrix} uv \\ v^2 \\ vw \end{bmatrix}, \quad \mathrm{G}_a = \begin{bmatrix} uw \\ vw \\ w^2 \end{bmatrix}, \\
&\mathrm{E}_v = \frac{1}{Re}\begin{bmatrix} u_x \\ v_x \\ w_x \end{bmatrix}, \quad \mathrm{F}_v = \frac{1}{Re}\begin{bmatrix} u_y \\ v_y \\ w_y \end{bmatrix}, \quad \mathrm{G}_v = \frac{1}{Re}\begin{bmatrix} u_z \\ v_z \\ w_z \end{bmatrix}, \\
&\mathrm{E}_p = \begin{bmatrix} p \\ 0 \\ 0 \end{bmatrix}, \quad \mathrm{F}_p = \begin{bmatrix} 0 \\ p \\ 0 \end{bmatrix}, \quad \mathrm{G}_p = \begin{bmatrix} 0 \\ 0 \\ p \end{bmatrix}, \\
&E_c = cu, \quad F_c = cv, \quad G_c = cw, \\
&E_d = \frac{1}{Re\,Sc}c_x, \quad F_d = \frac{1}{Re\,Sc}c_y, \quad G_d = \frac{1}{Re\,Sc}c_z
\end{aligned} \tag{4}$$

where, u, v, and w are velocity component of x, y, and z direction respectively, p is pressure, and c is concentration. The subscript x, y and z indicate derivatives with respect to x, y, and z respectively. Here, Re and Sc are Reynolds number and Schmidt number respectively. On the other hand, combined motion of the translation and rotation of a body is considered. The rigid body equations of motion are written as follows:

$$\frac{\mathrm{d}p_{\mathrm{B}}}{\mathrm{d}t} = \mathrm{f}_{\mathrm{B}}, \tag{5}$$

$$\frac{\mathrm{dL_B}}{\mathrm{d}t} = \mathrm{N_B}. \tag{6}$$

here, p_{B} is the momentum vector of the body, f_{B} is the external force vector, L_{B} is the angular momentum vector, and N_{B} is the torque vector, respectively.

2.2. Moving-grid finite-volume method

To assure the geometric conservation laws, we adopt a control volume in the space-time unified domain (x, y, z, t), which is four-dimensional in the case of three-dimensional flows. Now, Eq. (2) can be written in divergence form as,

$$\tilde{\nabla} \cdot \tilde{\mathcal{F}} = 0 \tag{7}$$

where,

$$\tilde{\nabla} = \begin{bmatrix} \dfrac{\partial}{\partial x} \\ \dfrac{\partial}{\partial y} \\ \dfrac{\partial}{\partial z} \\ \dfrac{\partial}{\partial t} \end{bmatrix}, \quad \tilde{\mathcal{F}} = \begin{bmatrix} E_a + E_p - E_v \\ F_a + F_p - F_v \\ G_a + G_p - G_v \\ q \end{bmatrix}. \tag{8}$$

The present method is based on a cell-centered finite-volume method and, thus, the flow variables are defined at the center of the cell in the (x, y, z) space. The control volume becomes a four-dimensional polyhedron in the (x, y, z, t)-domain, as schematically illustrated in Fig. 1.

We apply volume integration to Eq. (7) with respect to the control volume illustrated in Fig. 1. With use of the Gauss theorem, Eq. (7) can be written in surface integral form as,

$$\int_{\tilde{\Omega}} \tilde{\nabla} \cdot \tilde{\mathcal{F}} \mathrm{d}\tilde{V} = \oint_{\partial\tilde{\Omega}} \tilde{\mathcal{F}} \cdot \tilde{\mathrm{n}}_u \mathrm{d}\tilde{S} = \sum_{l=1}^{8} \left(\tilde{\mathcal{F}} \cdot \tilde{\mathrm{n}}\right)_l = 0. \tag{9}$$

here, $\tilde{\mathrm{n}}_u$ is an outward unit vector normal to the surface, $\partial\tilde{\Omega}$, of the polyhedron control volume $\tilde{\Omega}$ and, $\tilde{\mathrm{n}}_l = (\tilde{n}_x, \tilde{n}_y, \tilde{n}_z, \tilde{n}_t)_l$, ($l = 1,2,\ldots 8$) denotes the surface normal vector of control volume and its length equals to the boundary surface area in four-dimensional (x,y,z,t) space.

The upper and bottom boundary of the control volume ($l = 7$ and 8) are perpendicular to t-axis, and, therefore they have only $\tilde{n}_t$ component and its length is corresponding to the volume of the cell in the (x,y,z)-space at time t^n and t^{n+1} respectively. Thus, Eq. (9) can be expressed as,

$$\mathrm{q}^{n+1}(\tilde{n}_t)_8 + \mathrm{q}^n(\tilde{n}_t)_7 + \sum_{l=1}^{6} \left(\tilde{\mathcal{F}}^{n+1/2} \cdot \tilde{\mathrm{n}}\right)_l = 0. \tag{10}$$

In addition, Moving-Grid Finite-Volume method is applied to Eq. (3). Thus, Eq. (3) can be expressed as,

$$c^{n+1}(\tilde{n}_t)_8 + c^n(\tilde{n}_t)_7 + \sum_{l=1}^{6} \left(\tilde{\mathcal{F}}_c^{n+1/2} \cdot \tilde{\mathrm{n}}\right)_l = 0 \tag{11}$$

where,

$$\tilde{\mathcal{F}}_c = \begin{bmatrix} E_c - E_d \\ F_c - F_d \\ G_c - G_d \\ c \end{bmatrix}. \tag{12}$$