Fully explicit computational method for gas-solid two-phase flow with large temperature variation

Daisuke Toriu[1,*] and Satoru Ushijima[1]

[1]Academic Center for Computing and Media studies, Kyoto University

*toriu.daisuke.8v@kyoto-u.ac.jp

Abstract. This study proposes a fully explicit computational method for gas-solid two-phase flows with large temperature variations. To efficiently accelerate the fully explicit computation of non-isothermal low-Mach-number flows, we use a fractional step method with the reduced speed of sound technique (FSM-RSST). In addition, mechanical interactions between gas and solid phases are calculated by the direct-forcing/fictitious domain (DF/FD) method. The developed method is applied to the sedimentation of single cylinder. For the isothermal case, the time history of the sedimentation velocity is good agreement with reference results. Furthermore, results of the non-isothermal case demonstrate that the temperature distribution significantly affects on the sedimentation velocity of the cylinder.

Keywords: Fully explicit scheme, Non-isothermal flow, Fluid-solid interaction, Fictitious domain method

1. Introduction

Non-isothermal gas-solid two-phase flows are widely observed in technological processes and natural phenomena. Under small temperature variations, the Boussinesq approximation is usually adopted when computing such flows by assuming that the fluid is incompressible. However, the method which solves governing equations for compressible fluids is required to accurately calculate flows with large temperature variations because the fluid density varies significantly depending on the temperature.

In our previous studies [1, 2], a fractional step method with the reduced speed of sound technique (FSM-RSST) was proposed for explicitly computing low-Mach-number flows with a wide range of temperature variations. To overcome the strict limitation on the time increment due to the Courant–Friedrichs–Lewy (CFL) condition based on the speed of sound, one free parameter is introduced in the computation of the pseudo pressure, which is used in the momentum and energy equations. This parameter flexibly reduces the speed of sound depending on flow conditions, and FSM-RSST enables us to easily accelerate the fully explicit computation of compressible low-Mach-number flows.

The main objective of this study is the application of FSM-RSST to gas-solid two-phase flows by introducing the direct-forcing/fictitious domain (DF/FD) method [3], which is the method to compute fluid-solid interactions on the Cartesian coordinate. The developed method is applied to the sedimentation of single cylinder in the ideal gas with the large temperature variation. The calculated sedimentation velocity of the cylinder is compared with that obtained in the isothermal case.

2. Numerical method

The governing equations for compressible fluids used in this study consist of the mass conservation equation, momentum equations, and energy equation. The fluid is assumed to be an ideal gas, having the physical properties of air and constant specific heats. The solid is assumed to be a rigid body and its physical properties are constant. The pseudo pressure p' [1] used in the momentum and energy equations is defined as $p' = p_0 + \zeta\tilde{p} + (1-\zeta)\bar{P}_{\mathrm{f}}$, where $\tilde{p}$ is the pressure fluctuation from the initial pressure p_0, ζ is a time-varying but spatially constant parameter to reduce the speed of sound that satisfies $0 < \zeta \leq 1$, and $\bar{P}_{\mathrm{f}}$ is the spatially averaged value of $\tilde{p}$. The pseudo speed of sound can be controlled according to the value of ζ, and it in low-Mach-number flows is $\sqrt{\zeta}C$, where C is the the original speed of sound. This means that the proposed RSST enables us to use about $1/\sqrt{\zeta}$ times larger time increment compared with that in the non-RSST case ($\zeta = 1$).

The numerical procedure of FSM-RSST is divided into three stages [1]: computational stages of time evolution due to advection, diffusion, and other (pressure and external force) terms. The governing equations are discretized basically using the finite volume method on the staggered grid system. In addition, this study deals with two-dimensional cases. The free parameter ζ and the time increment Δt are updated at each time step as follows:

$$\zeta = \left[R_U \max\left(\frac{U_{\mathrm{r}}}{C}\right)\right]^2, \ \Delta t = N_{\mathrm{CFL}} \min\left(\min_{i=1,2} \frac{\Delta x_i}{|u_i| + \sqrt{\zeta}C}\right). \tag{1}$$

where U_{r} is the reference velocity, and N_{CFL} is the maximum Courant number. The notation "max" and "min" with no subscript represent the taking of the spatial maximum and minimum values. The parameter R_U represents the minimum value of the ratio of the $\sqrt{\zeta}C$ to U_{r}. To reduce the speed of sound without significantly affecting the computational accuracy, the value of R_U should be set sufficiently larger than 1.

Thermal interactions between fluid and solid phases are considered by solving the phase averaged heat conduction equation in the diffusion stage [2], and mechanical interactions between them are calculated by DF/FD method [3]. In DF/FD method, the fluid computations are mainly conducted on Eulerian grids. On the other hand, fluid-solid interaction force terms are estimated on the Lagrangian points arranged in the solid body, and they are transferred to Eulerian grids.

3. Sedimentation of single cylinder in non-isothermal gas

The proposed method is applied to the sedimentation of the single cylinder in the ideal-gas. Figure 1 shows the computational domain. Prior to the numerical experiment for the non-isothermal case, the benchmark test case [4] is calculated, in which all boundaries are adiabatic no-slip walls and the fluid is almost isothermal through the computation. The initial temperature T_0 of the fluid is 300 [K], the reference pressure P_0 is 8.61×10^7 [Pa], the reference fluid density ρ_f is 1000 [kg/m^3], the solid density ρ_s is $1.25\rho_f$, the viscosity μ is 0.01 [Pa·s], and gravitational acceleration g is 9.81 [m/s^2]. The specific heats at constant pressure and at constant volume of the fluid, c_p and c_v, are 1004.5 [J/(kg·K)] and 717.5 [J/(kg·K)], respectively. The specific heat c_s of the solid is 717.5 [J/(kg·K)]. The thermal conductivity λ_f of the fluid is determined so that the Prandtle number is equal to 0.71, and the thermal conductivity λ_s of the solid is the same as that of the fluid. The other conditions are as follows: the diameter d_p of the cylinder is 2.5×10^{-3} [m], the lengths L_1 and L_2 of the computational domain are $8d_p$ and $24d_p$, the initial coordinates X_{p1} and X_{p2} of the central point of the cylinder are $L_1/2$ and $2L_2/3$, the number of the computational cells is 240×720, and the number of Lagrangian points is 331. As for Eq. (1), we use $U_r = \sqrt{gd_p(\rho_s - \rho_f)/\rho_s}$, $R_U = 50$, and $N_{CFL} = 0.40$.

Figure 2 shows calculated vertical velocities of the cylinder and reference results by Wan and Turek [4]. The results in the RSST case are in good agreement with those in the non-RSST case ($\zeta = 1$) and the reference results. Here, ζ in the RSST case is about 1.0×10^{-4} through the computation. This indicates that Δt in the RSST case is about 100 times larger than that in the non-RSST case.

The computational conditions of the non-isothermal case are considered as follows: $T_0 = 400$ [K], $P_0 = 1.01325 \times 10^5$ [Pa], $\rho_f = 0.88$ [kg/m^3], $\rho_s = 1.40\rho_f$. To treat the gas with large temperature variations, μ_f and λ_f are determined with the Sutherland law. In the non-isothermal case, top and bottom boundaries are the no-slip walls having constant temperatures $T_h = 400$ [K] and $T_c = 300$ [K]. The other conditions are the same as those of the benchmark test case. Here we note that the sedimentation of the cylinder starts after the temperature distribution becomes steady under the above conditions.

Figure 3. shows the calculated temperature distributions. The sedimentation of the cylinder in the gas with the large temperature variation is stably calculated by the proposed method. The vertical velocity of the cylinder is compared between isothermal and non-

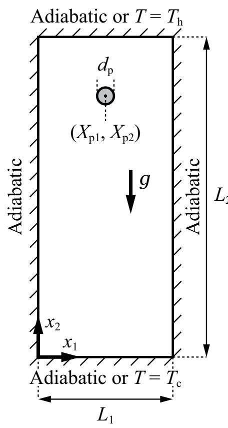

Figure 1: Computational domain

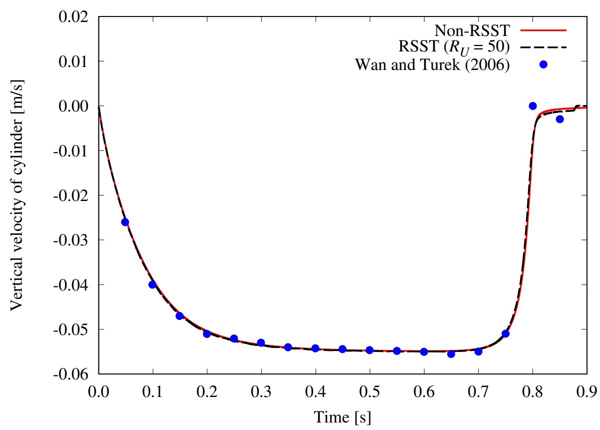

Figure 2: Calculated vertical velocities of the cylinder and reference results [4] for the benchmark test case

isothermal cases in Fig. 4. Here we note that “isothermal case” indicates the case in which we set $T_h = T_c = 400$ and other conditions are the same as those in the non-isothermal case. The results shown in Fig. 4 demonstrates that the temperature distribution significantly affects on the motion of the cylinder. In the isothermal case, the sedimentation velocity of the cylinder becomes almost constant at $0.6 \leq t \leq 0.8$ [s], and then it decreases sharply when the cylinder approaches the bottom wall. On the other hand, in the non-isothermal case, the sedimentation velocity decreases before the cylinder approaches the bottom wall because the gas density and viscosity become larger toward the bottom depending on the temperature distribution. These results indicate the importance of using the compressible fluid model for the computations of gas-solid two-phase flows with large temperature variations.

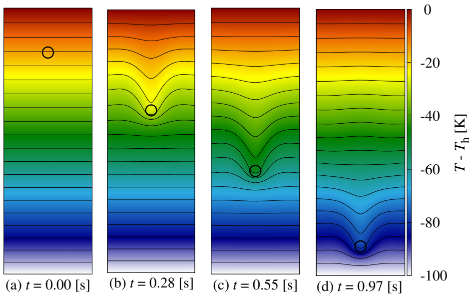

Figure 3: Temperature distribution

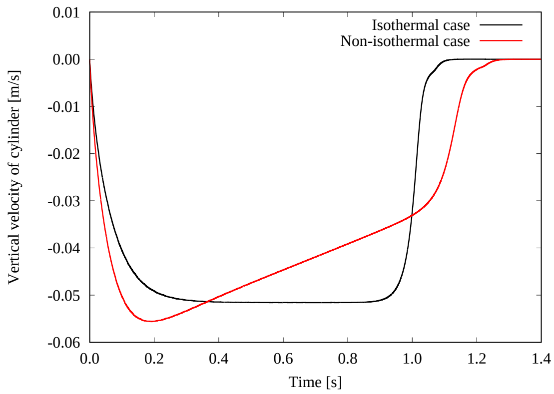

Figure 4: Comparison of vertical velocities between isothermal and non-isothermal cases

Acknowledgements

This work was supported by JSPS KAKENHI Grant Number JP19K20284 and a research grant from the Mazda Foundation.

References

[1] D. Toriu, S. Ushijima: Computation of non-isothermal and compressible low Mach number gas flows by fully explicit scheme using control method for speed of sound *Journal of Advanced Simulation in Science and Engineering,*, 6:1 (2019), 260–272.

[2] D. Toriu, S. Ushijima: Computations of thermal interactions between natural convection with high temperature difference and solid objects based on reduced speed of sound technique *The 34th Computational Fluid Dynamics Symposium*, (2020), D04-2.

[3] D. Nie, J. Lin: A LB-DF/FD method for particle suspensions *Communications in Computational Physics*, 7:3 (2010), 544–563.

[4] D. Wan, S. Turek: Direct numerical simulation of particulate flow via multigrid FEM techniques and the fictitious boundary method *International Journal for Numerical Methods in Fluids*, 51:5 (2006), 531–566.