International Journal of Heat and Mass Transfer 120 (2018) 284–299

Contents lists available at ScienceDirect

International Journal of Heat and Mass Transfer

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

Direct numerical simulation of flow around a heated/cooled isolated sphere up to a Reynolds number of 300 under subsonic to supersonic conditions

Takayuki Nagata [a,*], Taku Nonomura [a], Shun Takahashi [b], Yusuke Mizuno [c], Kota Fukuda [d]

[a] Department of Aerospace Engineering, Tohoku University, 6-6-01, Aramaki, Aoba-ku, Sendai, Miyagi 980-8579, Japan
[b] Department of Prime Mover Engineering, Tokai University, 4-4-1, Kita-kaname, Hiratsuka, Kanagawa 259-1292, Japan
[c] Course of Science and Technology, Tokai University, 4-4-1, Kita-kaname, Hiratsuka, Kanagawa 259-1292, Japan
[d] Department of Aeronautics and Astronautics, Tokai University, 4-4-1, Kita-kaname, Hiratsuka, Kanagawa 259-1292, Japan

ARTICLE INFO

Article history:
Received 8 August 2017
Received in revised form 21 November 2017
Accepted 9 December 2017

Keywords:
DNS
Temperature effect
Spherical particle
Compressible flow
Particle drag coefficient

ABSTRACT

In this study, an analysis of the flow properties around an isolated sphere under isothermal conditions for flows with high Mach numbers and low Reynolds numbers is conducted via direct numerical simulation (DNS) of the three-dimensional compressible Navier–Stokes equations. The calculations are performed with a boundary-fitted coordinate system. The Reynolds number based on the diameter of the sphere and the freestream quantities is varied from 100 to 300, the freestream Mach number is varied between 0.3 and 2.0, and the temperature ratio between the sphere surface and the freestream is varied between 0.5 and 2.0. We focus on the effects of the Mach number and the temperature ratio on the flow properties. The results show the following characteristics: (1) unsteady vortex shedding from the sphere is promoted (suppressed) when the temperature ratio is less (greater) than unity; (2) the drag coefficient increases with the temperature ratio, but previous drag relations give poor prediction on effect of the temperature ratio on the drag coefficient in the continuum regime; (3) Nusselt number relations proposed in previous studies can be applied if the temperature ratio is close to unity under subsonic conditions; (4) the changes in several flow properties can be characterized by a separation point in the range investigated.

1. Introduction

The exhaust gas from rocket engines generates strong acoustic waves that could cause critical damage to the payload in the fairing. Therefore, it is very important to be able to predict and reduce the acoustic level at liftoff. Traditionally, the acoustic level has been predicted semi-empirically using either NASA SP-8072 [1] or subscale tests [2]. However, NASA-SP8072 is based on numerous launch data obtained in the United States and so is not suitable as a design tool for new launch pads and rockets. Recently, studies of acoustic level prediction using computational fluid dynamics (CFD) have been conducted. Tsutsumi et al. [3,4] performed simulations of the acoustic field considering the effect of the launch facility and the flame deflector plate. In addition, Nonomura et al. [5] conducted an analysis considering the difference in the specific heat ratio of the exhaust jet and the atmosphere. As a result, acoustic phenomena have been examined with limited accuracy.

The exhaust jet contains small particles such as aluminum droplets released from the solid rocket motor or water droplets introduced by water injection during the launch of large liquid-propellant rockets. From experimental results, it is known that these particles attenuate acoustic waves; however, the mechanism of this process has not been explained. In addition, the effect of the particle is not simple to scatter the sound [6,7]. Therefore, highly accurate predictions of the acoustic level require the particle influences to be examined. To increase the accuracy of acoustic predictions using CFD, more detailed aspects of the complex physics should be taken into account. However, the aluminum droplets released from solid rocket motors are approximately 1–200 μm in diameter [8] and the exhaust gas flow is supersonic. Therefore, the flow around each particle experiences conditions of a high Mach number M and a low Reynolds number Re (e.g., when particles passing through the shock wave).

In a general gas–particle flow analysis, the particles are approximated as point masses. Therefore, interactions between the particles and the fluid are considered using the drag and Nusselt number relations for particles. Several particle drag relations for

* Corresponding author.
E-mail address: nagata.takayuki@aero.mech.tohoku.ac.jp (T. Nagata).

https://doi.org/10.1016/j.ijheatmasstransfer.2017.12.042
0017-9310/

compressible flows have been proposed [9–13]. These are based on theoretical formulas, empirical expressions and corrections, and experimental data obtained under limited conditions. However, the applicability of these relations has not been examined. Fig. 1 shows a map of previous studies on the drag coefficient of a sphere under compressible and low-*Re* conditions. Here, *Kn* in Fig. 1 denotes the Knudsen number defied as the ratio of mean free-path length of the surrounding molecules to the diameter of the sphere.

For example, measurements of the drag coefficient using sonic-speed micron-size particles and a Faraday cage were performed by Crowe et al. [14]. In that investigation, particles were accelerated by a static electric field, and the particle deceleration was measured using a Faraday cage. The particle drag coefficient was estimated from the particle deceleration. Zarin and Nicholls [15] obtained drag data for *Re* = 40–500, *M* = 0.1–0.57, Knudsen numbers as high as 0.06, and turbulence intensities up to 13% (their presented data were only for *Re* = 40–200 and *M* = 0.17–0.57). Bailey and Hiatt [16] conducted free-flight measurement teste of the drag coefficient of a sphere at subsonic to hypersonic speeds under continuum to nearly free-molecular conditions using a ballistic range. In addition, Macrossan [17] provided direct simulation Monte Carlo (DSMC) drag data from a rarefied regime.

From these experimental and numerical studies, the qualitative characteristics of the drag coefficient are clear except in limited regions. However, the accuracy of some of the experimental data is limited by the severe conditions under which the experiments and measurements were performed. In addition, the characteristic of the flow field and other properties have not been examined sufficiently. Moreover, the temperature dependence of the flow properties has not been examined for compressible flows. Various researchers have proposed Nusselt number relations of spherical object [18–20]. For example, Sauer [19] proposed a Nusselt number relation by extending the theoretical expression of heat transfer for flat plates, and Ranz and Marshall [18] and Fox et al. [20] proposed a Nusselt number relation by estimating the amount of heat transfer in experiments on the evaporation of liquid droplets and the shock wave ignition of magnesium powders, respectively.

Various researchers have examined numerically the flow properties of incompressible flows around an isolated sphere at low-*Re* flows. For example, Johnson and Patel [21] analyzed numerically and experimentally the flow around a sphere up to *Re* = 300 under incompressible flows. They examined the *Re* dependence of the flow geometries, hydrodynamic force coefficients, types of flow pattern, and vortical structures. In addition, Kurose et al. [22] investigated the flow properties of a heated/cooled sphere for *Re* = 50–400 in incompressible flows, examining the effect on the flow properties of the ratio of the temperature of the sphere to that of the freestream. Kajishima and Takiguchi [23] suggested influences of the wake vortices released from the particles on the formation of particle clusters in the incompressible flow regime via a particle-resolved direct numerical simulation (DNS) using the immersed boundary method. However, the energy dissipation of the flow field because of the interaction of the wake vortices released from the particles and the turbulence, and the effect of the wake vortices on the particle distribution cannot be considered that in the multiphase flow models constructed by the drag and Nusselt number relations. Therefore, an examination of the flow properties such as the drag coefficient, the Nusselt number, and flow structures of the sphere is essential for constructing a highly accurate multiphase flow model. Accordingly, we have performed DNS of the flow around a sphere under high-*M* and low-*Re* conditions to construct a multiphase flow model that includes the influence of the particles. In previous work, we investigated the flow properties and applicability of previous drag relations [24], and we suggested that

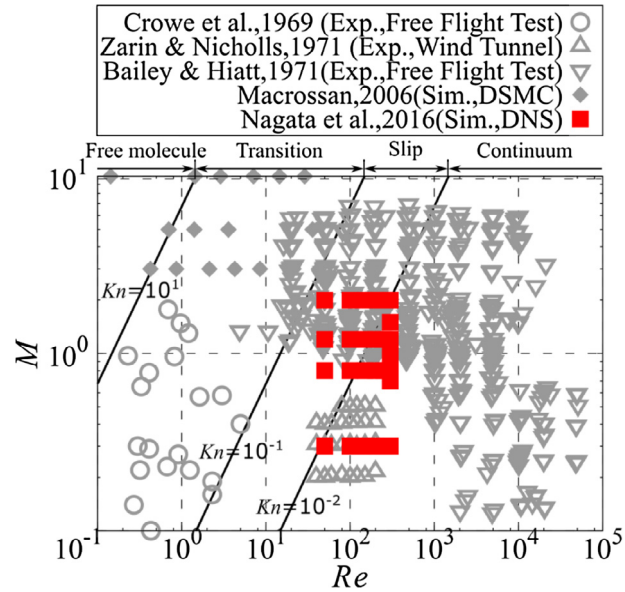

Fig. 1. Map of previous studies of the drag coefficient of a sphere under compressible and low-*Re* conditions.

the previous relations are not sufficiently accurate at high-*M* and low-*Re* conditions. We believe that it is necessary to examine the effect of the particle/freestream temperature difference on the flow properties under high-*M* and low-*Re* conditions to improve the multiphase flow model of a rocket exhaust jet. In the present study, we examine the influences of a sphere/freestream temperature difference via an analysis of the isothermal conditions at the surface of the sphere.

2. Methodologies

2.1. Governing equations

The three-dimensional compressible Navier–Stokes equations are employed as the governing equations. The Navier–Stokes equations in Cartesian coordinates are as follows:

$$\frac{\partial Q}{\partial t}+\frac{\partial E}{\partial x}+\frac{\partial F}{\partial y}+\frac{\partial G}{\partial z}=\frac{\partial E_v}{\partial x}+\frac{\partial F_v}{\partial y}+\frac{\partial G_v}{\partial z}, \tag{1}$$

where Q contains conservative variables, E, F, and G are the x, y, and z components of an inviscid flux, and E_v, F_v, and G_v are the x, y, and z components of a viscous flux. We have that

$$\begin{aligned} Q &= (\rho \quad \rho u \quad \rho v \quad \rho w \quad e)^T, \\ E &= (\rho u \quad \rho u^2+p \quad \rho u v \quad \rho u w \quad (e+p)u)^T, \\ F &= (\rho v \quad \rho v u \quad \rho v^2+p \quad \rho v w \quad (e+p)v)^T, \\ G &= (\rho w \quad \rho w u \quad \rho w v \quad \rho w^2+p \quad (e+p)w)^T, \end{aligned} \tag{2}$$

$$\begin{aligned} E_v &= (0 \quad \tau_{xx} \quad \tau_{xy} \quad \tau_{xz} \quad \beta_x)^T, \\ F_v &= (0 \quad \tau_{yx} \quad \tau_{yy} \quad \tau_{yz} \quad \beta_y)^T, \\ G_v &= (0 \quad \tau_{zx} \quad \tau_{zy} \quad \tau_{zz} \quad \beta_z)^T, \end{aligned} \tag{3}$$

$$\begin{aligned} \beta_x &= \tau_{xx}u+\tau_{xy}v+\tau_{xz}w-q_x, \\ \beta_y &= \tau_{yx}u+\tau_{yy}v+\tau_{yz}w-q_y, \\ \beta_z &= \tau_{zx}u+\tau_{zy}v+\tau_{zz}w-q_z. \end{aligned} \tag{4}$$