International Journal of Multiphase Flow 61 (2014) 1–13

Contents lists available at ScienceDirect

International Journal of Multiphase Flow

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

Unsteady effects in dense, high speed, particle laden flows

J.D. Regele [a,*], J. Rabinovitch [b], T. Colonius [c], G. Blanquart [c]

[a] Aerospace Engineering, Iowa State University, Ames, IA 50011, USA
[b] Graduate Aerospace Laboratories, California Institute of Technology, Pasadena, CA 91125, USA
[c] Mechanical Engineering, California Institute of Technology, Pasadena, CA 91125, USA

ARTICLE INFO

Article history:
Received 9 May 2013
Received in revised form 24 December 2013
Accepted 29 December 2013
Available online 9 January 2014

Keywords:
Dense particle cloud
Shock wave
Particle laden
Drag coefficient

ABSTRACT

Dense high speed non-compacted multiphase flows exist in variable phase turbines, explosions, and ejector nozzles, where the particle volume fraction is in the range $0.001 < \alpha_d < 0.5$. A canonical problem that can be used to study modeling issues related to these types of flows is a shock wave impacting a planar particle cloud. Thus far, prior work has modeled the flow using a 1-D volume-averaged point particle approach and developed momentum and energy coupling terms that reproduce accurately the trajectory of particles in the experiments. Although these early results are promising, it is appropriate to question whether all aspects of the experimental flow can be captured using a one-dimensional model that is traditionally only used for dilute flows. Thus the objective of this work is to set-up a two-dimensional configuration that captures qualitatively the multidimensional behavior of a real three-dimensional particle cloud, but can be used as an exact solution to compare with an equivalent volume-averaged model. The 2-D data is phase-averaged to reduce it to one dimension, and x–t diagrams are used to characterize the flow behavior. These results show the importance of the Reynolds stress term inside the particle cloud and in its turbulent wake. A one-dimensional (1-D) model is developed for direct comparison with the 2-D simulation. While the 1-D model characterizes the overall steady-state flow behavior well, it fails to capture aspects of the unsteady behavior inside and behind the particle cloud because it neglects important unclosed terms.

1. Introduction

Dense high speed multiphase flows can be found in a variety of practical applications such as variable phase turbines, explosions, liquid rocket motors, and ejectors. In these applications, particles can be spaced far apart or close together. The range of scales encompassed by these applications precludes direct numerical simulation in the design process. Thus, sub-grid scale models are necessary in order to model these flows at practical scales.

There are different regimes of multiphase flow based on the disperse phase volume fraction α_d. In the dilute regime $\alpha_d \ll 1$, there are no collisions because the particles are far apart and the particles do not interact with each other. In the limit of the granular regime $\alpha_d \sim 1$, the particles are in constant contact with each other; their motion is controlled primarily by particle collisions, with small contributions from the continuous phase. The regime of interest in this paper is the dense non-granular regime ($0.001 < \alpha_d < 0.5$) between these two limiting extremes (Zhang et al., 2001).

Experiments and simulations used to study these regimes often rely on an initial shock wave to set up a high-speed flow through a particle cloud. In the dilute regime, these flows are well characterized (Carrier, 1958; Miura and Glass, 1982, 1983; Miura, 1990; Saito, 2002). Similarly, in the granular regime where the particles are densely packed, continuum mixture theories exist to describe these flows (Baer and Nunziato, 1986; Powers et al., 1990). However, little information exists for particle flow interactions where the volume fraction is in the range $0.001 < \alpha_d < 0.5$. Recently, a multiphase shock tube experiment has been developed at Sandia National Labs (Wagner et al., 2011, 2012) to explore dense high speed particle-laden flow. This experiment improves upon other configurations by isolating the dense non-compacted regime with a gravity fed curtain of nearly constant volume fraction in the stream-wise direction.

In the dilute regime, for both low Mach number (incompressible) and compressible flows, volume-averaged models combined with a point-particle approach have been used successfully to model multiphase flows (Clemins, 1988; Drew, 1983; Drew and Passman, 1999; Magnaudet and Eames, 2000; Crowe et al., 2012). These models rely on the assumption that the particle diameters are much smaller than the inter-particle distance. Under these conditions, the particles interact with the continuous phases independent from other particles and the coupling terms between the

* Corresponding author. Tel.: +1 515 294 1614; fax: +1 515 294 3262.
E-mail addresses: jregele@iastate.edu (J.D. Regele), jason.rabinovitch@caltech.edu (J. Rabinovitch), colonius@caltech.edu (T. Colonius), g.blanquart@caltech.edu (G. Blanquart).

0301-9322/$ - see front matter © 2014 Elsevier Ltd. All rights reserved.
http://dx.doi.org/10.1016/j.ijmultiphaseflow.2013.12.007

continuum and disperse phases can be modeled as independent single particle interactions (Crowe et al., 2012).

The point-particle approach, which assumes the individual particles are independent, is not valid for the granular regime where the flow is collision dominated. In other words, granular flow may not be modeled using information for only a single particle. However, in the dense non-compacted regime ($\alpha_d < 0.5$), this distinction is not so clear because the collisions are less frequent and the continuous phase contributes significantly to the particle dynamics. Simultaneously, the particles are sufficiently far apart that their effect on the continuum phase *might be* represented accurately enough using the same source terms as for the dilute regime.

Recently Ling et al. (2012) combines the experimental data of Wagner et al. (2012) with a one-dimensional volume-averaged point-particle model including unsteady momentum coupling forces. The leading and trailing edges of the particle cloud and transmitted and reflected shock locations predicted by the numerical model accurately match the experimental data. This was done by incorporating correction functions for finite particle Mach number and volume fraction. The correction function for Mach number is developed for a single sphere (Parmar et al., 2010) and the correction function for finite volume fraction is developed assuming the fluid has only minor acoustic waves (Sangani et al., 1991). In the current work, it will be demonstrated that volume-averaged models can reproduce accurately the reflected and transmitted shock locations, but that this approach incorrectly accounts for the large amount of energy that is contained in turbulent structures.

The goal of the present paper is to investigate the interactions between disperse and continuum phases in the dense non-compacted regime. More specifically, the objectives are threefold: (1) assess the validity of the volume-averaged models combined with a point-particle approach to model compressible multiphase flows; (2) identify the limitations/shortcomings of this technique; and (3) propose modifications to this approach that will better capture the physical phenomena that occur in these types of flows. To streamline the discussion, we ignore heat and mass transfer so that the results are restricted to a disperse phase of solid, adiabatic particles.

Towards this goal, the canonical problem of a shock wave impacting a planar particle cloud is considered. This configuration is intended to mimic the physics encountered in the aforementioned investigation of Wagner et al. (2012): namely compressible flow, dense non-compacted disperse phase, and nearly planar configuration (on average). While the coupling between the two phases is in reality two-way, the present work focuses only on the early stages of the experiments where the particles can be assumed, with good accuracy, to remain fixed in space. This should be considered as a necessary first step towards understanding the physics of the fluid-particle interaction.

The paper is organized as follows. In the following section, a 2-D configuration is presented that will model a planar shock wave impacting a particle cloud alongside the numerical framework used to solve the governing equations. Then, in Section 3, the numerical results of the 2-D simulations are presented. In Section 4, the phase-averaged equations for the present configuration are rederived and commonly made assumptions are summarized and discussed. Finally, in Section 5, the solution of the 1-D phase-averaged equations are compared with the phase-averaged results of the 2-D exact flow field.

2. Detailed simulation of shock-particle interaction

In this section, a numerical simulation that captures the fluid-particle interactions for a dense multiphase flow is developed.

2.1. Assumptions

In the experiment of Wagner et al. (2012), the Reynolds number based on the velocity behind the reflected shock and the particle diameter is approximately $Re \approx 2000$. At this Reynolds number, the drag coefficient of a single individual particle is nearly constant and the flow is considered to be in the inertial range (Clift et al., 1978). Within the inertial range, viscous forces on the particles are negligible, and the drag force on a particle is predominantly from pressure drag. Nevertheless, it is important to note that molecular viscosity (albeit very small) is the reason for the separation of the boundary layer, which is why the drag coefficient is constant. In the present work, the Euler equations are used to model the flow through the particle cloud. As will be shown later, the artificial viscosity associated with the numerical discretization is sufficient to induce boundary layer separation and a realistic flow around the particles.

A two-dimensional simulation will be used to characterize the flow behavior. Although true multiphase flow is three-dimensional, two-dimensional simulations are an efficient first step in extracting the qualitative physical behavior. The flow around a sphere (3-D) and a cylinder (2-D), particularly the boundary layer separation point and transition Reynolds numbers, is different. However, the 2-D simulations are sufficient to investigate the fundamental differences between direct simulation and a volume-averaged point-particle model. A full three-dimensional demonstration will be the subject of future work.

The final major assumption in this work is that the particles have infinite inertia and can be considered frozen in place. While this assumption is perfectly valid for the objectives of this work, this limits the comparison that can be made with the experimental data (Wagner et al., 2012). Fig. 1 shows three different Schlieren

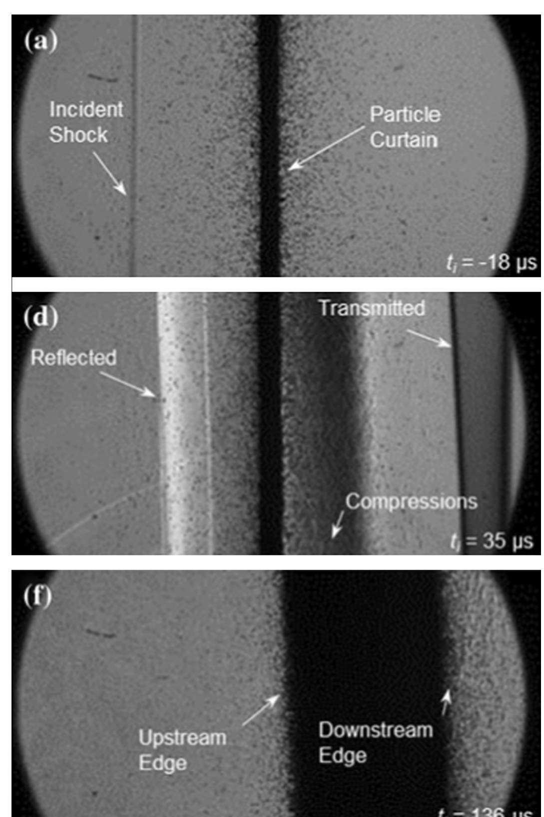

Fig. 1. Selected Schlieren images at times −18, 35, and 136 μs from Wagner et al. (2012).