Available online at www.sciencedirect.com
ScienceDirect
International Journal of Heat and Fluid Flow 28 (2007) 1136–1143
International Journal of HEAT AND FLUID FLOW
www.elsevier.com/locate/ijhff

# Numerical analysis of shock wave interaction with a cloud of particles in a channel with bends

Pawel Kosinski *

*The University of Bergen, Department of Physics and Technology, Allegaten 55, 5007 Bergen, Norway*

Received 7 August 2006; received in revised form 4 November 2006; accepted 9 November 2006
Available online 23 January 2007

**Abstract**

Shock wave interactions with a cloud of particles were analysed using an Eulerian–Lagrangian numerical technique. In this approach, the particle–particle and particle–wall interactions can be modelled directly. Simulations were carried out in a channel with two bends: an issue important for pneumatic and slurry transport of particles, as well as for dust explosions in industrial facilities. At the beginning, the results are shown as snapshots, showing the reader how the particle cloud propagates through the channel. At the next stage some statistics is performed in order to analyse the influence of channel geometry. The following parameters are analysed: the history of the percentage of particles that have travelled along the whole channel and the history of the average kinetic energy of the particles. It is shown how the channel structure and particle diameter influences the parameters.


*Keywords:* Shock waves; Particle/fluid flows; Numerical simulation; Particle transport; Dust explosions

## 1. Introduction

### 1.1. Overview of recent literature

In literature there are a few dozen publications devoted to the topic of shock wave interaction with a cloud of particles, sometimes in domains with complex geometry. Applications range from dust explosions in coal mines and grain elevators, to solid propellant combustion in rocket engines. Both experimental and numerical simulation techniques have been used in these publications and detailed analyses of shock wave and particle behaviour have been performed. Here a few papers are mentioned as examples illustrating a general trend.

Wang et al. (2001) present numerical simulations of shock wave behaviour in the presence of a square cavity filled with particles. The authors state that investigation of shock wave interaction with both obstacles and particles is important for understanding complex phenomena, such as dust explosions in industrial facilities. The simulation is in a two-dimensional domain, and the gas is assumed to be inviscid. The simulation is performed both for pure gas and the gas with particles, and a detailed comparison is made. The authors emphasize that numerical simulations of such processes may be a research tool, which is superior to experiments. Experiments are often complex and not easy to repeat. The same has been mentioned in many other papers as well.

Park and Baek (2003) perform simulations of the interaction of a shock wave with a cloud of carbon particles that either undergo combustion or are inert. Additionally, a wedge is located in the computational domain to make the process more complex and thus more real. The combustion is modelled as a simple one-step heterogeneous reaction between carbon and oxygen.

Igra et al. (2004) analyse shock wave reflection from a wedge where the whole domain is filled with a dusty gas. Different solids loadings and particle diameters are considered. This paper differs from other works where only part of the computational domain is filled with dust, often near

* Tel.: +47 55 58 28 17; fax: +47 55 58 94 40.
*E-mail address:* Pawel.Kosinski@ift.uib.no

0142-727X/$ - see front matter 
doi:10.1016/j.ijheatfluidflow.2006.11.003

the wedge. In the latter type of simulations, the shock wave initially propagates in a pure gas and later interacts with a particle cloud, giving rise to an extra phenomenon, namely reflection of the wave from the cloud, and this phenomenon Igra et al. try to avoid.

Boiko et al. (1997) pay special attention to the drag force coefficient. They point to the well-known fact that, in spite of great number of data in literature, this coefficient is only well determined as a function of the Reynolds number and only for a single particle. The Mach number needs to be taken into account for high relative velocities, as does the particle concentration for flows with high concentration. In the first part of their paper, Boiko et al. present some experimental results, where the value of the drag force coefficient is determined, and later they use this in numerical simulations. The results of computation are validated against experiments.

A complex and interesting problem is the interaction of a shock wave with a layer of particles. Since the boundary layer is turbulent, the vortex structures scoop dust out of the bed of particles and the dust is then entrained up and over the vortex structures and is then transported by the turbulent flow. This was shown, e.g. by Kuhl et al. (1989) and Collins et al. (1994). Another explanation is the particle–particle interactions (Kosinski et al., 2005; Kosinski and Hoffmann, 2006).

In Klemens et al. (2001), results of simulations are presented where the geometry of a real channel with fixtures and fittings is mimicked by including vertical obstacles. The interphase interaction is described by simple relations for drag force and convective heat exchange, and additionally the Magnus lift force is implemented. In the paper particle–particle interactions are not taken into account, even though such interactions are likely to dominate for the high particle concentrations present in dust layers.

Kosinski et al. (2005) compare two numerical techniques: Eulerian–Eulerian and Eulerian–Lagrangian. The former has been frequently used for analysis of dust lifting behind shock wave, in spite of the fact that modelling of particle–particle and particle–wall collisions may be questionable using this approach. Therefore Kosinski et al. suggest using the Eulerian–Lagrangian technique and they show that this leads to superior results. Also in Kosinski and Hoffmann (2005) a similar issue is addressed.

Rogue et al. (1998) study experimentally and numerically the interaction of a shock wave with a bed of particles. The bed is situated in a vertical tube and fluidization is induced by a shock wave travelling from the bottom of the tube forcing the particles to move upwards. Very detailed experimental analysis is performed, where the dust bed consists of one, two and more layers of particles. Thanks to that, different phenomena could be included or excluded in the numerical simulations (e.g. collisions that are not important for only one layer). Rogue et al. also experimentally determine the drag force coefficient, which they later use for their numerical investigation. Finally, the numerical results are compared with the experiments.

Also an interesting problem is the interaction with a dust layer, where the chemical reaction between the particles and an oxidiser in the gas, is taken into account. This has been especially emphasized in works: Fedorov and Gosteev (2002), Gosteev and Fedorov (2003), Fedorov and Fedorchenko (2005). See also works: Rose et al. (1999, 2000).

The problem of shock wave propagation in a channel becomes complex, when the channel is not straight but consists of bends and obstacles. This process is even more challenging to analyse when the flow is two-phase. The solid particles are subject to the interaction with the flowing gas, as described in detail in the papers mentioned above, but also with channel walls. This is of importance for studying pneumatic or slurry of particles, but also for such issues like dust explosions. Especially for modelling of dust explosions, the problem of particle–particle and particle–wall interactions is often neglected in order to simplify the computer code and the analysis process. Nevertheless, in the author's previous paper: Kosinski (2006) a problem of dust mixture propagation in a branched channel is described. Two simulation techniques are compared there: Eulerian–Eulerian and Eulerian–Lagrangian. It is shown that modelling particle–wall collisions is crucial when the channel geometry is complex.

In this paper the following problem is considered: propagation of dust mixture in a channel with two bends after having interacted with a shock wave. The Eulerian–Lagrangian modelling technique is used in order to investigate such processes. The results are shown as snapshots, showing the reader how the particle cloud propagates through the channel, but also some statistics is performed in order to analyse the influence of channel geometry and particle size.

### *1.2. Simulation technique*

Two techniques are generally used for simulating two-phase flow phenomena: Eulerian–Eulerian (E–E) and Eulerian–Lagrangian (E–L) (see e.g. Crowe et al., 1998).

In the E–E approach both phases are treated as separate fluids, coupled by, for instance, drag force and heat exchange. This approach is computationally economical and suitable for dense mixtures with two-way coupling, but it requires good phenomenological models, usually based on kinetic theories for granular flows. It is especially challenging to implement phenomena like particle–particle and particle–wall interactions, even though attempts at doing this have been reported in literature.

In the E–L approach, the particles are tracked in the computational domain and in this aspect this kind of model is more physically correct. Also, it is much straightforward to implement phenomena such as particle–particle and particle–wall interactions, as well as some particle-fluid interactions, such as lift forces.

The objective of this paper is to perform simulations of the interaction of a shock wave with a cloud of particles in a channel with two bends using the E–L approach.