Available online at www.sciencedirect.com
ScienceDirect
Powder Technology 186 (2008) 189 – 198
POWDER TECHNOLOGY
www.elsevier.com/locate/powtec

A comprehensive approach in modeling Lagrangian particle deposition in turbulent boundary layers

M. Horn [a,*], H.-J. Schmid [b]

[a] Evonik Degussa GmbH, Rodenbacher Chaussee 4, 63457 Hanau, Germany
[b] Mechanical & Environmental Process Engineering, University of Paderborn, 33098 Paderborn, Germany

Received 1 April 2007; received in revised form 7 September 2007; accepted 24 November 2007
Available online 19 February 2008

Abstract

Modeling of particle deposition on adjacent walls is a key issue in various applications like separation or transport processes. The present paper focuses on the modeling of turbophoretic deposition of particles in the micron size range. The first step is to evaluate the important range where turbophoresis plays an important role in comparison to other mechanisms e.g. gravity or electrostatic separation. The disadvantages of commonly used models will be analyzed and overcome by implementing a more sophisticated approach considering damping of turbulent fluctuations in the wall-boundary layer. In contrast to previous work, commonly used turbulence models are applied to solve the mean flow field of the examples under consideration. The results will show a good prediction of particle deposition in comparison to experimental values [B.Y.H. Liu, J.K. Agarwal, Experimental observation of aerosol deposition in turbulent flow, Aerosol. Sci. 5 (1974) 145–155.] by using the advanced model.

Keywords: Euler; Lagrange; Modeling; Turbulence; Boundary layer; Deposition

1. Introduction

Modeling of the near wall treatment within the simulation of two-phase flows plays an important role in many practical applications. The focus on this near wall treatment is necessary if either desired or undesired transport of the particulate phase to the wall is relevant. Typical examples are deposition in duct or pipe flows (most probably undesired deposition), dust precipitation or coating processes (desired deposition). Numerical simulation of turbophoretic separation of fine particles, i.e. particle deposition in case of turbulent confined flows, is the key issue of this paper.

For the following, a two-phase flow is defined as combination of continuous phase, e.g. gas or liquid, and disperse phase, e.g. particles or droplets. Small liquid droplets ($d_P < 50$ μm) are assumed to behave physically like solid particles [2]. Two-phase flows comprised by a continuous and a disperse phase are commonly simulated by the Euler–Lagrange approach in commercial CFD-Codes: The continuous phase is solved by the Eulerian approach whereas the disperse phase is treated as discrete particle tracks by the Lagrangian approach [3]. The k–ε-model is widely used for the simulation of turbulent fluid flows in practical applications. The important part is the coupling between continuous and disperse phase, especially the influence of the turbulent characteristics. Many approaches are successfully used within the core flow of the continuous phase [4,5]. However, a lack of information exists in the near wall region.

The following topics will be discussed within this paper: When does turbophoresis play an important role in such simulations? How do commonly used commercial codes deal with the lack of information in the near wall region? Finally, a more accurate approach is presented in the fine particle regime using a statistical turbulence model solved using a commercial CFD code and a particle tracking model with special boundary layer treatment. This approach will be discussed and applied to model the given tasks.

The challenge in predicting the deposition of small particles suspended in a turbulent gas flow has been investigated for several years. The first experimental and modeling studies have

* Corresponding author.
E-mail address: michael.mh.horn@evonik.com (M. Horn).

0032-5910/$ - see front matter © 2007 Elsevier B.V. All rights reserved.
doi:10.1016/j.powtec.2007.11.048

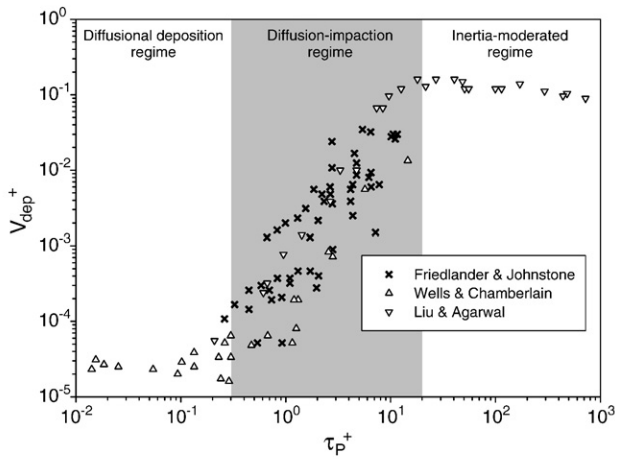

Fig. 1. Summary of experimental data on particle deposition from fully developed pipe flow: Friedlander & Johnstone [6], Wells & Chamberlain [9] and Liu & Agarwal [1].

been reported by Friedlander [6]. A summary of experimental results is published by Young and Leeming [7] or Matida et al. [8]. The overview given in Fig. 1 reports the results obtained by Friedlander & Johnstone [6], Wells & Chamberlain [9] as well as Liu & Agarwal [1]. The deposition rate on the wall of a cylindrical pipe is expressed in terms of the deposition velocity V_{dep} (Eq. (1)). The term is normalized with the friction velocity u_τ. The term is a characteristic value for turbulent flow fields accounting for the wall shear stress τ_W at given flow conditions and will be discussed within the theoretical section (Eq. (14)).

$$V_{dep}^{+} = \frac{V_{dep}}{u_\tau} = \frac{1}{u_\tau}\frac{\dot{V}_{fl}}{A_{dep}} \ln \frac{\dot{N}_{in}}{\dot{N}_{in} - \dot{N}_{dep}} \quad (1)$$

The definition is based on a balance for a finite volume section and expressed in terms of the flow rate of the continuous fluid $\dot{V}_{fl}$, the area of the deposition section in this section A_{dep} as well as inlet particle rate $\dot{N}_{in}$ as well as deposited particle rate $\dot{N}_{dep}$. The plots for the deposition velocity are given for different particle diameters d_P and fluid flow conditions expressed by the particle relaxation time τ_P (Eq. (2)) in dimensionless form. The terms include the Cunningham correction Cu, the dynamic fluid viscosity η_{fl} as well as particle density ρ_P and the fluid density ρ_{fl}.

$$\tau_P^{+} = \tau_P \frac{\tau_W}{\eta_{fl}} = \frac{d_P^2 \rho_P Cu}{18\eta_{fl}} \frac{\tau_W}{\eta_{fl}} \quad (2)$$

Obviously, three different regimes can be identified [7]. The inertia-moderated regime is relevant for large particles ($\tau_P^{+} > 20$), i.e. a particle size above ($d_P > 30$ μm) for the present conditions of the fluid flow. The particles in this size range acquire a wall normal component of the velocity by the turbulence of the core flow. The particles have too much kinetic energy to be affected by the boundary layer. Consequently, those particles are separated very much. This regime is not considered, because it is state of the art in simulation tools to predict the size range very good.

Very small particles ($\tau_P^{+} < 0.3$) are not only affected by fluid fluctuations but also by Brownian motion. Very small separation velocity is observed, as those particles follow the core flow nearly ideally. The particles in the vicinity of a wall are then separated by the stochastic Brownian motion [10].

The paper is focused on the simulation of the transition between diffusional deposition and inertia-moderated regime: Diffusion impaction regime. The small particles follow but not ideally the turbulent structures within the boundary layer. Therefore, the paper will focus on the comparisons of numerical study with the experimental data obtained by Liu and Agarwal. Those results are representative and cover the compete range of interest.

How is it possible to model this certain range of interest and solve the particle deposition for practical applications? Several attempts have been made to investigate this task with help of a combination of analytical solutions, Eulerian approaches or DNS data for the mean flow field and different approaches for the particle transport within boundary layer [11,12]. In practice, the focus is on Euler–Langrange approaches, i.e. solving the mean flow field based on a continuous approach and the dispersed phase by discrete particle tracking. This approach is competitive for many applications, if a local resolution is important or complex models are applied on the physics of the particulate phase. No satisfying result is obtained so far, when applying a statistical turbulence model and a Lagrangian approach on this specific topic of turbulent particle deposition within the considered particle size range.

2. Modeling

2.1. Particle motion

The Lagrangian approach for the simulation of the disperse phase is based on Newton's equation of motion (Eq. (3)). The particles are assumed as rigid, spherical and inert. In case of bubbles or droplets internal convection or shape oscillations might occur which could be accounted for by an approach given by Feng [13] or Crowe et al. [14], respectively. In case of non-spherical particles the drag force would have to be modified accordingly. Furthermore, particle rotation might become relevant as well giving rise to additional forces (like e.g. Magnus force). However, very little is known about the influence of particle rotation of non-spherical particles. Therefore we must restrict these investigations to spherical particles, which is a common assumption for small particles and will have no significant influence on the deposition characteristics.

$$m_p \frac{d\vec{u}_p}{dt} = \vec{F}_D(\vec{u}, \vec{u}_p) + \vec{F}^{\,i} \quad (3)$$

The left hand side accounts for inertia. The first term on the right hand side represents the drag force and the second one represents additional forces (e.g. gravity, Coulomb force, lift forces etc.). The drag force depends on the instantaneous fluid velocity u (Eq. (6)) and the instantaneous particle velocity $\vec{u}_p$. The coupling between fluid and particle velocity is essential for the turbulent deposition of particles. Gravity can be neglected for small particles. Even though some publications found a considerable influence of gravity in vertical pipe flows [8], the