



Experimental study and numerical investigation of particle penetration and deposition in 90° bent ventilation ducts

Hai Jiang, Lin Lu*, Ke Sun

Department of Building Services Engineering, The Hong Kong Polytechnic University, Hung Hom, Kowloon, Hong Kong SAR, China

ARTICLE INFO

Article history:

Received 18 February 2011

Received in revised form

21 April 2011

Accepted 27 April 2011

Keywords:

Particle-gas flow

90° bend

Penetration

Deposition

CFD simulation

ABSTRACT

Particle deposition in a ventilation duct is a severe problem as it can pose health hazards. This is more evident in 90° bent ventilation ducts. The exposure of particle deposition location in bend section is important and useful in understanding and dispelling particle contamination. This paper investigated particle penetration and deposition in 90° bent ventilation ducts numerically and validated using experimental and previous research data. In the numerical study, particle penetration and deposition in a 2D 90° bend turbulent flow were analyzed. The Renormalized Group (RNG) $k-\epsilon$ model and Lagrangian particle tracking model were utilized to characterize turbulent gas flow and particle behavior, respectively. Particle turbulent dispersion was introduced by adopting the eddy lifetime model with the near wall fluctuating velocity corrected to take turbulence anisotropy into account. In the experimental validation, particle pollution collected from an actual ventilation duct was observed. The particle penetration rates in a test duct at 6 different Stokes numbers were measured for validation. The numerical results were consistent with both the experimental study and the data obtained from previous research.

© 2011 Elsevier Ltd. All rights reserved.

1. Introduction

As people spend more and more time in the indoor environment nowadays, the particle pollution in a ventilation system can be a major threat to human health [1]. Better understanding of the particle distribution and deposition status in ventilation ducts is thus necessary and important to help improving indoor air quality.

Inside the HVAC system, the bend section is commonly used to connect straight ducts and change flow direction. Considering the complex circumstances in turbulent bend flows, experimental analysis of the particle behavior becomes quite challenging. The velocity distributions of airflow and particle flow in bends were firstly focused by earlier experimental studies (Kliafas and Holt [2], Yang and Kuan [3]). Following that, the penetration behavior of a particle through 90° bends was reported by Pui [4], McFarland et al. [5] and Peters and Leith [6]. From the limited previous experimental studies, it is found that the particle penetration rate has a relationship with Stokes number in the 90° bends. Recently, Sippola [7] investigated particle deposition velocity in the bend section and found that particle deposition in turbulent bend flows

was often greater than in straight ducts for equal particle sizes and air speeds.

Nowadays, computational fluid dynamics (CFD) has become an efficient and robust tool to reveal the particle phenomena in turbulent flows, and several numerical studies on particles in bend flows by using CFD were reported. Berrouk and Laurence [8] performed a well-resolved LES to compute the deposition efficiency of aerosols in a turbulent circular cross-section bend flow. Particles with Stokes number that ranges between 0.005 and 1.5 were concerned. Breuer et al. [9] predicted the aerosol deposition in 90° bends using LES and an efficient Lagrangian tracking method and they found the particles can closely follow the continuous flow for low Stokes numbers. Tian et al. [10] employed the Lagrangian model, together with a particle-wall collision model, to study the effects of wall roughness on particle rebound behavior. Our previous work [11] studied the particle distribution and concentration in a 90° bend flow by incorporating a particle-wall impact model. In the numerical studies of particle deposition in turbulent air flow, the Renormalization Group (RNG) $k-\epsilon$ model was usually employed to solve indoor duct turbulent airflow due to its relative simplicity and robustness [12]. Meanwhile, the particle phase was normally solved by the discrete trajectory approach (Lagrangian method), which had been successfully validated in the previous studies as reported in [13–15].

* Corresponding author. Tel.: +852 3400 3596; fax: +852 2765 7198.

E-mail address: bellu@polyu.edu.hk (L. Lu).