Journal of Wind Engineering & Industrial Aerodynamics 179 (2018) 385–406

Contents lists available at ScienceDirect

Journal of Wind Engineering & Industrial Aerodynamics

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

Numerical study of fluid flow and particle dispersion and deposition around two inline buildings

H.R. Haghighifard [a], M.M. Tavakol [a,*], G. Ahmadi [b]

[a] Department of Mechanical Engineering, Shiraz Branch, Islamic Azad University, Shiraz, Iran
[b] Department of Mechanical and Aeronautical Engineering, Clarkson University, Potsdam, NY, USA

ARTICLE INFO

Keywords:
Supported buildings
CFD
Turbulent flow
Surface roughness
Deposition fraction
Lagrangian tracking

ABSTRACT

In the present study, turbulent airflows and particle deposition around two inline buildings were investigated numerically. A computational modeling approach including the RNG k-ε and Realizable k-ε turbulence models were used for these simulations. The Lagrangian particle tracking approach was implemented for evaluating dispersion and deposition of spherical dust particles. For simulation of turbulence fluctuations, an improved discrete random walk (DRW) model that includes the near wall correction was implemented into several user-defined functions (UDFs) that were linked to the ANSYS-Fluent code. It was shown that the new improved DRW stochastic model led to results that are more accurate compared to the standard model. The improved DRW model was then used for simulating turbulent deposition and dispersion of dust around building models. The presented results showed that putting buildings on elevated supports reduces dusts deposition from downstream of single and inline buildings at short distances, particularly for small particles of about 1 μm. It is also shown that particle deposition fractions around buildings on rough ground are higher than those for smooth ground. Deposition fractions were also predicted by the improved model for faces of single and inline buildings.

1. Introduction

Scientists and engineers have been concerned with particle deposition in internal and external flows due to their significant industrial and environmental applications. Recently, there has been considerable interest in the dust transport and deposition processes in HVAC ducts and around buildings due to their importance in environmental air pollution and the associated health issues. In last three decades, there have been several studies on dust transport and deposition processes over smooth and rough flat plates (Sehmel, 1971; Wedding and Stukel, 1974; Clough, 1975; Paz et al., 2013). Experimental measurements of particle deposition in duct flows were reported by Friedlander and Johnstone (1957), Rosinski and Langer, 1967 and Liu and Agarwal (1974) among others. They found that particle deposition rate increases with airflow speed. Wood (1981), Hinds (1984) and Papavergos and Hedley (1984) provided extensive reviews of theoretical and experimental studies of transport and deposition of particles in turbulent flows. McLaughlin (1989), Ounis et al. (1993), Zhang and Ahmadi (2000), and Li et al. (2016) studied turbulent flows and particle deposition in duct flows using the direct numerical simulation (DNS).

Understanding the features of flow and pollutant transport near building models has also attracted considerable attention in the recent years. An earlier experimental study on dispersion of plume near cubical building model was reported by Robins and Castro (1977). Pesava et al. (1999) discussed the nature of separated flows and sub-micron aerosol deposition on a cube in cross winds. They observed that the deposition rates on various cube faces are different. Liu and Ahmadi (2006) investigated helium gas and particle transport and deposition around a building model. Flow field simulations were performed with the RSTM turbulence model and particle tracking was done by use of the Lagrangian approach. Their results indicated large particle are deposited mainly by impaction on the side of the building facing the wind, but the smaller particle deposition rate are more uniform. They noted that gravity effects are significant for particles larger than 10 μm, and Brownian motion was important for aerosol particles smaller than 0.1 μm. Nazridoust and Ahmadi (2006) simulated the flow and pollutant transport in street canyons. They used the Langrangian approach for evaluating the solid particle dispersion and deposition. In investigation of pollutant dispersion around an urban building model, Zhang et al. (2015) found that concentration of haze fog is not relevant to position of emission. Ai and Mak (2014) and Yu et al. (2017) investigated particle dispersion around multistory building models at different angle of wind

* Corresponding author.
E-mail address: tavakolmm@shirazu.ac.ir (M.M. Tavakol).

https://doi.org/10.1016/j.jweia.2018.06.018
Received 5 April 2018; Received in revised form 25 June 2018; Accepted 26 June 2018

incident.

Moshfegh et al. (2010) studied particle deposition and dispersion around two inline 2D cylinders numerically. Particles were released from various source and the corresponding deposition efficiencies were evaluated. Wevers and Hoffer (2012) investigated particle dispersion around two buildings. Comparison between their numerical simulations and experiments showed that the prediction of particle concentration behind isolated building is overestimated, however, the concentration at the back of building with an open hall is underestimated. Tominaga and Stathopoulos (2017) compared steady Reynolds Average Navier-Stokes simulations (SRANS) and unsteady RANS (URANS) models for predicting flow field and pollution concentration around a cubic building model. They reported that URANS was more successful than SRANS model for prediction of concentration when the release source was located at the back of the building model.

In the present study, the flow field and particle deposition around single supported and inline standard (non-supported) and supported low-rise buildings for three streamwise distances were studied. The results for single standard building were also presented for comparison. The simulated airflow fields around single and inline buildings were validated by comparison with the available experimental data. The predicted micron-size particle deposition fraction on the ground around single and inline buildings was validated by calculation of deposition velocity over smooth and rough flat plates and comparing them with the experimental results. In addition, the deposition fractions on various faces of standard and supported building models were investigated and compared with earlier experimental and numerical results and available theories. Then, the effect of surface roughness on the velocity distribution and deposition fraction of particles on the ground around buildings was evaluated and discussed.

2. Geometry and boundary conditions

In this paper, airflow and particle dispersion and deposition around scaled models of single supported building and two inline buildings were simulated numerically. The cases of standard surface-mounted buildings, and when the buildings were supported on open frames were considered. Here a scale factor of 0.01 with respect to the full-scale buildings was used. With this scale factor, for a standard building, a 10 cm long, 10 cm wide and 3 cm height cube was considered. The supported building model had the same dimensions, but was mounted on four 0.4 cm × 0.4 cm × 1 cm supports. Inline building models were positioned at different streamwise distances of S=H, 2H and 5H with H = 3 cm was the standard building model height.

Selecting proper boundary conditions are critical to the accuracy of Computational Fluid Dynamic (CFD) results. In recent years, certain guidelines for selection of boundary conditions for simulation of flow around buildings and urban areas were recommended in the literature (Franke et al., 2007; Tominaga et al., 2008). In the present study, the Working Group of the Architectural Institute of Japan (AIJ) guidelines (Tominaga et al., 2008) are used for appropriately modeling the flow field around single and inline buildings. The AIJ guideline suggests use of atmospheric boundary layer velocity profile for inlet wind flow, generation of appropriate computational domain to avoid blockage and side way interference and developing proper meshing of the computational domain around building. In addition, recommendations are provided for improving convergence of the numerical solutions and use of proper turbulence models. The developed computational domain is shown in Fig. 1 and the imposed boundary conditions at different planes are listed in Table 1.

The developed computational grids consisted of 1.6 million hexahedral grid cells for the single building models. Grid quality assessment shows that the maximum aspect ratios for standard and supported building models were, respectively, about 8 and 10. For the inline buildings, the computational grids were in the range of 2–2.2 million cells. Number of grids for length (x direction), width (y direction) and height (z direction) around the single building models were 152 × 72 × 60 for the single building models.

Fine grids were generated around the buildings and near wall. Fig. 2-a and 2-b show sample computational grid sections around inline standard surface-mounted and supported building models at the symmetry plane (Y/H = 0) for separation distance of S=H. Structure of grids around supports is shown in Fig. 2-c. Grid sensitivity tests were performed and the results are discussed in Section 7.

The AIJ guideline was used and proper velocity distribution, turbulence kinetic energy and dissipation rate profiles at the inlet of compu-

Table 1
Boundary conditions used.

Planes in Fig. 1	Boundary condition
abcd, a'b'c'd'	Velocity inlet
efgh, e'f'g'h'	Pressure outlet
abfe,a'b'f'e'	Wall
cghd, c'g'h'd'	Symmetry
adhe,a'd'h'e'	Symmetry
bcgf, b'c'g'f'	Symmetry

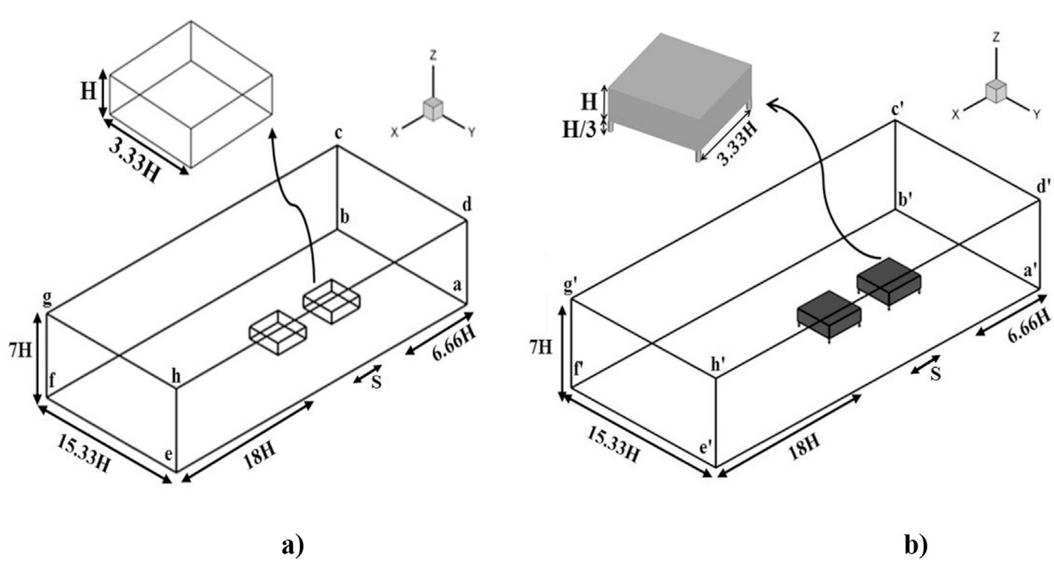

Fig. 1. Computational domain for inline buildings, a) Standard buildings, b) Supported buildings.