Cold Regions Science and Technology 151 (2018) 109–118

Contents lists available at ScienceDirect

Cold Regions Science and Technology

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

LES and wind tunnel test on friction velocity on roof surfaces

Xuanyi Zhou[a], Gongbo Zu[b,a,*], Ming Gu[a], Jinhai Hu[a]

[a] State Key Laboratory of Disaster Reduction in Civil Engineering, Tongji University, Shanghai 200092, China
[b] Department of Civil Engineering, The University of Hong Kong, Pokfulam Road, Hong Kong

ARTICLE INFO

Keywords:
Snow saltation
Friction velocity
Flat roof
LES
Wind tunnel test

ABSTRACT

Snow saltation is governed by the shear stress from wind on the snow surface, in form of the friction velocity. In most previous studies, the friction velocity was measured or estimated by wind velocity at a certain height using the log law, which is suitable for the fully developed wind field of an open area. However, the applicability of such method to roofs has not been tested. In this study, Large Eddy Simulation (LES) of wind flow around a flat roof was performed to study the flow field characteristics above a flat roof and the results were compared with those from the wind tunnel measurement. Owing to the flow separation and the reversed flow above the roof, the log law is observed to exist only on the leeward part of the roof; therefore, measurement based on the log law is not universally suitable to estimate the friction velocity on roofs. The comparison of the simulation results against experimental data shows that LES reasonably reproduces the flow field around the roof and can be an alternative approach to estimate the friction velocity on building roofs. The fluctuating friction velocity is found to be unneglectable compared with the mean friction velocity and should be considered in the estimation of the snow transport on roofs.

1. Introduction

In the field of civil engineering, snow redistribution by wind transport may lead to uneven snow distribution on roofs during a snowstorm and could ultimately collapse roof structures. A considerable amount of economic losses due to snow accumulation has been reported (Ross, 1984; Rens, 2000). Thus, accurate prediction of snow redistribution on roof surface is critical to structural design. The wind-induced transport of snow is generally regarded to be governed by the wall shear stress acted on snowpack surfaces (Pomeroy and Gray, 1990). In the study of snow drifting, the wall shear stress between wind flow and snow surface, which is typically written in the form of friction velocity, is measured by the Irwin probe and other instruments in wind tunnel experiments (Preston, 1953; Irwin, 1981; Walter et al., 2014; Ferreira et al., 2015). In real environment, the friction velocity is usually calculated from the measured wind velocity of the fully developed boundary layer using the log law (Pomeroy and Gray, 1990). However, the wind field around a bluff body, such as a building roof, is much complicated and spans of roofs are insufficiently long; thus, the fully developed boundary layer over roofs may not be formed for most cases. Hence, accurate measurement or estimation of the wall shear stress or the friction velocity is important to study the snowdrift on roofs and would be a challenging topic.

With the development of computers, since the 1990s, researchers began to employ computational fluid dynamics (CFD) to study the snow drifting phenomenon (Uematsu et al., 1991; Liston and Sturm, 1998; Sundsbo, 1998; Beyers et al., 2004; Tominaga et al., 2008, 2011; Thiis and Ramberg, 2008; Ferreira et al., 2015; Huang and Wang, 2016; Huang et al., 2016; Faria et al., 2017; Wang and Huang, 2017). The friction velocity is still the key parameter in the estimation of snow redistribution in CFD simulations. In addition, the friction velocity was calculated by the simulated wind speed near snow surfaces using the log law in the preceding studies mentioned. When the CFD method is adopted to simulate snow drifting on roof surfaces, a similar problem would also occur. The roof span is usually insufficiently long for the formation of a fully developed boundary layer; thus the wall shear stress could dramatically vary along the roof span for flow separation. Therefore, the log law method may be unsuitable for predicting the friction velocity of roofs with limited span. In addition, the turbulence modeling of previous numerical simulations all relied on Reynolds Averaged Navier-Stokes (RANS) method (e.g., Zhou et al., 2016a; Ferreira and Thiis, 2017), in which obtaining the fluctuating component of the wind field is difficult. However, for building roofs, the wind flow is somewhat turbulent since roofs are usually located at the bottom of the atmospheric boundary layer. The fluctuating component of the friction velocity and its effects on snow transport on roofs have to be carefully evaluated when the CFD method is adopted.

Therefore, the present study adopts large eddy simulation (LES) to

* Corresponding author at: Department of Civil Engineering, The University of Hong Kong, Pokfulam Road, Hong Kong
E-mail address: gbzu@hku.hk (G. Zu).

https://doi.org/10.1016/j.coldregions.2018.03.005
Received 17 September 2017; Received in revised form 30 January 2018; Accepted 6 March 2018
Available online 15 March 2018

simulate the friction velocity on roof surfaces and discusses how its unsteady characteristics influences snow transport on roofs. First, the LES method was applied to study the dynamic wind flow field around a flat roof. Then, the mean and the fluctuating wind velocities above the roof were measured in a wind tunnel test. The flow characteristics between the simulated results from the LES method and those obtained from the wind tunnel test were compared to validate the simulated numerical results. Finally, the distributions of the time-averaged mean and fluctuating friction velocities on the roof surface and its possible influence on the snow transportation are discussed.

2. Simulation of shear stress and friction velocity on roofs

2.1. Governing equations

The numerical simulations in the present study were performed using commercial CFD software Ansys Fluent. In the LES turbulence model, large-scale eddies are explicitly resolved by solving the filtered Navier–Stokes equations whereas only small eddies are modeled. The governing equations of LES are obtained by filtering the time-dependent Navier–Stokes equations as follows:

$$\frac{\partial \rho \tilde{u}_i}{\partial x_i} = 0 \tag{1}$$

$$\frac{\partial}{\partial t}(\rho \tilde{u}_i) + \frac{\partial}{\partial x_j}(\rho \tilde{u}_i \tilde{u}_j) = \frac{\partial}{\partial x_j}\left(\mu \frac{\partial \tilde{u}_i}{\partial x_j}\right) - \frac{\partial \tilde{p}}{\partial x_i} - \frac{\partial \tau_{ij}}{\partial x_j} \tag{2}$$

where $\tilde{u}_i$ and $\tilde{p}$ are the filtered mean velocity and the filtered pressure respectively, ρ and ν are the air density and the dynamic viscosity, respectively, and τ_{ij} is the subgrid-scale stress which is modeled as follows:

$$\tau_{ij} = -2\mu_t \tilde{S}_{ij} + \frac{1}{3}\tau_{kk}\delta_{ij} \tag{3}$$

$$\tilde{S}_{ij} \equiv \frac{1}{2}\left(\frac{\partial \tilde{u}_i}{\partial x_j} + \frac{\partial \tilde{u}_j}{\partial x_i}\right) \tag{4}$$

where μ_t is the subgrid-scale turbulent viscosity, and $\tilde{S}_{ij}$ is the rate-of-strain tensor for the resolved scale.

The Smagorinsky-Lilly model (Lilly, 1992) is used for the subgrid-scale turbulent viscosity, where the eddy viscosity is modeled as follows:

$$\mu_t = \rho L_s^2 \ |\tilde{S}| = \rho L_s \sqrt{2\tilde{S}_{ij}\tilde{S}_{ij}} \tag{5}$$

$$L_s = \min(\ \kappa\delta, C_s V^{1/3}) \tag{6}$$

where L_s is the mixing length for subgrid-scales, κ is the von Karman constant, δ is the distance to the closest wall and V is the volume of a computational cell. The dynamic version of the Smagorinsky model (Germano et al., 1991) was employed in the present study, and C_s is computed at each time step with a test-filter and clipped to the range of 0 to 0.23 to avoid numerical instabilities. This imposed maximum value of 0.23 for C_s follows the default value in Ansys Fluent and is found to be appropriate for flow around an isolated bluff body (Gousseau et al., 2013).

2.2. Friction velocity

In the study of snow drifting, the friction velocity u_τ is a very important parameter. This parameter is re-written from the wall shear stress τ in units of velocity and represents the shear force exerted by flow in motion on a solid boundary in the parallel direction (Schlichting and Gersten, 1999):

$$u_\tau = \sqrt{\tau/\rho} \tag{7}$$

where ρ is the air density.

Usually, the friction velocity u_τ, combined with the snow property represented as the threshold friction velocity u_{*t}, can determine whether erosion or accumulation occurs on the snowpack surface. Once the friction velocity u_τ exceeds the threshold friction velocity u_{*t}, snow saltation or drifting would occur and result in erosion. If the friction velocity u_τ is lower than the threshold friction velocity u_{*t}, then the snow would accumulate on the snow surface (Pomeroy and Gray, 1990; Zhou et al., 2016b).

Most of the previous studies on snow drifting aimed to explore the snow transport in open areas; thus, the friction velocity u_τ is generally calculated from the measured wind speed profile through the log law (Clifton et al., 2006; Pomeroy and Gray, 1990). This method requires a fully developed wind field to enable the vertical wind velocity profile to follow the log law of the atmosphere boundary layer. Thus, the wall shear stress on snow surface exerted by wind, which is represented by the friction velocity u_τ, can be calculated from the wind speed above. This method is certainly reasonable for an open field. However, when the study shifts its focus from the fully-developed wind profile to the flow pattern around a bluff-body, such as a flat roof, the flow phenomenon becomes complex due to flow separation and reverse flow. For such circumstances, the flow characteristics of the so-called "near-wall region" need to be carefully considered for the accurate calculation of the wall shear stress.

For the near-wall region, an essential condition is that the velocity on the wall surface is zero when the viscous fluid flows along the wall surface and the wall shear stress between the motion fluid and the rigid surface leads to a velocity gradient in the fluid in a direction normal to the flow. Thus, a layer establishes itself close to the wall, which is known as boundary layer. The region $y^+ \leq 100$ is usually considered to be the "near-wall region", which includes the laminar sublayer, the buffer region, and at least part of the logarithmic region (Robinson, 1991). The detailed turbulent structure of the boundary layer can be found in the literature (e.g., Saric et al., 2003).

In LES, the calculation of the wall shear stress, or the friction velocity, depends on the distance of the centroids of cells adjacent to the wall according to the boundary layer theory (Ansys Fluent 13.0, 2011). When a wall-adjacent cell is in the laminar sublayer, the friction velocity is obtained from the linear relationship as follows:

$$\frac{\tilde{u}}{u_\tau} = \frac{\rho u_\tau y}{\mu} = y^+ \tag{8}$$

If the mesh cannot resolve the laminar sublayer, then the centroid of the wall-adjacent cells falls within the logarithmic region of the boundary layer, and the law-of-the-wall is employed:

$$\frac{\tilde{u}}{u_\tau} = \frac{1}{\kappa}\ln E\left(\frac{\rho u_\tau y}{\mu}\right) \tag{9}$$

where $\tilde{u}$ is the filtered velocity tangential to wall; u_τ is the friction velocity, and the constant E is 9.793.

3. Research object and numerical procedure

3.1. Research object

Snow loads on a flat roof are important for most current load codes and standards. Thus, the research object of the present study is a flat roof with a height of 3.6 m and span of 14.4 m, and wind blows from left to right as shown in Fig. 1.

3.2. Computational domain and boundary conditions

The numerical simulations were conducted using a length scale of 1:30 with a roof dimension of length (L) × breadth (B) × height (H) = 48 cm × 48 cm × 12 cm to compare the simulated wind flow around the roof with that obtained from the wind tunnel experiment,