Urban Climate 9 (2014) 115–133

Contents lists available at ScienceDirect

Urban Climate

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

Flow simulations for simplified urban configurations with microscale distributions of surface thermal forcing

J.L. Santiago [a,*], E.S. Krayenhoff [b], A. Martilli [a]

[a] *Atmospheric Pollution Division, Environmental Department, CIEMAT, Avda. Complutense 40, Ed. 3, 28040 Madrid, Spain*
[b] *Department of Geography, University of British Columbia, Canada*

ARTICLE INFO

Article history:
Received 26 February 2014
Revised 21 July 2014
Accepted 29 July 2014

Keywords:
Aligned array of cubes
Drag coefficient
Realistic surface heat fluxes
Reynolds-averaged Navier–Stokes (RANS) simulations
Urban canopy

ABSTRACT

Effects of solar position and of the ratio of buoyant to inertial forces on flow properties (wind speed and temperature) within an array of urban-like obstacles are investigated. An aligned array of cubes with a plan area density of 0.25 is studied. First, distributions of sensible heat fluxes on built surfaces for different solar positions are determined by the Temperatures of Urban Facets in 3-D (TUF3D) model, a microscale three-dimensional urban energy balance model. These fluxes provide boundary conditions to a Reynolds-Averaged Navier–Stokes (RANS) model, which simulates flow and temperature over the array. Results are evaluated against experimental measurements. Furthermore, microscale variation of flow and temperature as a function of the ratio of buoyant to inertial forces is analyzed. Finally, spatially-averaged flow properties are determined as a function of this ratio, and inform neighbourhood-scale parameterizations for mesoscale modeling of urban flows. Differences between configurations are larger at the microscale than the neighbourhood (spatially-averaged) scale. Spatially-averaged flow properties are impacted primarily for scenarios with high buoyancy-to-inertial force ratios. In addition, an analysis of stationary vs. unsteady flow simulations is made.

* Corresponding author.
E-mail address: jl.santiago@ciemat.es (J.L. Santiago).

http://dx.doi.org/10.1016/j.uclim.2014.07.008
2212-0955/

1. Introduction

A more complete understanding of the micrometeorology and pollutant dispersion of built areas is useful for many applications, for example urban climate, air quality and pedestrian comfort. Interaction between atmospheric flow and urban elements (e.g. buildings and trees) induces complex flow patterns within the urban canopy and results in heterogeneous fields of temperature and pollutants. These obstacles act on the flow in two ways: (1) they block, deviate and slow the flow (mechanical effects); and (2) they exchange heat with the atmosphere, exerting buoyant forces on the flow (thermal effects).

Thermal effects on flow in street canyons have been studied by means of numerical simulations using Reynolds-averaged Navier–Stokes (RANS) models (Sini et al., 1996; Xie et al., 2006) or large eddy simulations (LES) models (Park et al., 2012; Cai, 2012). Evaluation of these numerical results with full scale measurements (e.g., Offerle et al., 2007; Louka et al., 2002; Rotach et al., 2005) has been problematic because forcing/boundary conditions are not easily controlled in the measurements. For this reason, the wind tunnel experiment conducted by Uehara et al. (2000), where the ground (street) between an array of cubes is heated, has been used for this purpose in similar studies (Xie et al., 2006; Park et al., 2012), because external conditions can be better controlled. These data are also used to evaluate our model configuration.

To date, microscale numerical flow studies for air quality purposes have either neglected thermal effects entirely, or accounted for them in a very crude way. Most scenarios studied have only heated one canyon wall, or the ground or one wall and the roof (Sini et al., 1996; Xie et al., 2006; Buccolieri et al., 2008; Park et al., 2012; Cai, 2012). However, for clear sky conditions, surface heat fluxes exhibit strong spatial heterogeneity due to building layout and orientation relative to the evolving solar angle. In this work, a realistic distribution of sensible heat fluxes at a resolution 1 m is computed for each scenario.

Effects of spatial variation of surface sensible heat flux on spatially-averaged flow properties are also investigated. Computational fluid dynamics (CFD) models explicitly resolve the flow around obstacles but their computational domains cannot cover the whole city because of the computational expense. As such, mesoscale models are necessary to simulate the whole city. Urban canopy parameterizations (UCPs) are developed for this purpose, and represent a compromise between simplicity and accuracy. In order to improve these parameterizations, spatially-averaged flow properties and drag coefficients obtained from CFD computations can be useful (Martilli and Santiago, 2007; Santiago and Martilli, 2010). There have been no studies (as far as the authors are aware) of the influence of thermal effects on the neighbourhood-scale drag coefficient, and this is investigated here.

The primary aim of this contribution is to assess the effects on flow properties (wind speed and temperature) of varying the solar position and the ratio between buoyancy and dynamical processes. First, the model is evaluated, and second, the analyses of microscale flow and temperature fields for the different cases are carried out. Third, spatially-averaged properties of the flow are investigated.

2. Numerical methods and set-up

The urban morphology is usually characterized in terms of packing density (plan area density) of the buildings, and the non-dimensional ratios λ_f and λ_p are used. λ_f is the ratio between the total frontal area of the obstacles and the total plan area, and λ_p is the ratio between the total plan area of the obstacles (or roofs) and the total plan area. In this case, an aligned array of cubes with $\lambda_f = \lambda_p = 0.25$, which is a typical packing density of real urban areas (Grimmond and Oke, 1999), is simulated by a computational fluid dynamics (CFD) model (STARCCM+, a commercial code from CD-Adapco, 2012). The simulations are steady state and conducted with a Reynolds-averaged Navier–Stokes (RANS) model that solves the standard k-ε turbulence equations and the energy conservation equation with temperature as the solved variable. In addition, buoyancy terms are accounted for with Boussinesq's approximation. This approach considers the air density constant except for the buoyancy term where changes in the density are expressed in terms of temperature (T), air density (ρ), thermal expansion coefficient (β) and gravitational vector ($\mathbf{g}$) as $\rho \mathbf{g} \beta \Delta T$. Details about these equations can be found in