Available online at www.sciencedirect.com

ScienceDirect

Solar Energy 99 (2014) 1–10

SOLAR ENERGY

www.elsevier.com/locate/solener

Fluid dynamic and heat transfer parameters in an urban canyon

S. Bottillo *, A. De Lieto Vollaro, G. Galli, A. Vallati

Sapienza University of Rome – DIAEE – Via Eudossiana 18, 00184 Rome, Italy

Received 28 June 2013; received in revised form 22 October 2013; accepted 24 October 2013
Available online 22 November 2013

Communicated by: Associate Editor Matheos Santamouris

Abstract

A microclimatic analysis in a typical urban configuration, has been carried out. Using a CFD method, a N-S oriented urban street canyon, with a given H/W ratio, has been examined. The standard k–ε turbulence model has been used to simulate a three-dimensional flow field and to calculate the thermo-fluid dynamics parameters that characterize the street canyon. The aim of this study is to investigate the effect of solar radiation on the flow field and thermal parameters within the canyon. A comparison between transient and stationary simulations has been performed to evaluate the importance of considering the thermal inertia effects in an urban street canyon study. The dynamic characteristics of the 3D flow in the canyon have been compared with other numerical simulations and experimental results. Furthermore a thermo-fluid dynamic analysis of natural convection effects on the heat transfer coefficient and turbulent kinetic energy, has been carried out.

Keywords: Urban microclimate; Urban canyon; CFD; Solar radiation

1. Introduction

The landscape of dense urban areas can be described by units of street delimited by two continuous rows of buildings to form a “canyon”. This geometry is often described by a single parameter, the canyon aspect ratio (H/W), which is defined as the ratio of the building height (H) to the width between buildings (W). As to the incoming solar radiation and the heating of canyon surfaces, the orientation of the canyon relative to the solar path is also critical in determining the timing and extent to which surfaces receive direct sunlight. Several studies have been performed on different street canyons (Takebayashi and Moriyama, 2012; Bozonnet et al., 2005; Lei et al., 2012; Xie et al., 2007). An experimental validation of a 3D numerical simulation has been performed by Assimakopoulos et al. (2006); they performed tests using a numerical model on a grid of buildings. By numerical tests characterized by a 2D spatial domain and with assigned surfaces temperatures, Lei et al. (2012) studied the impact of ground heating on the flow fields in a street canyons, Xie et al. (2007) studied the effects of façades and ground heating on the pollutant dispersion, Saneinejad et al. (2011) investigated on the heat transfer coefficient in a street canyon, simulated as a cavity, using the low-Reynolds number modeling; they found a strong influence of thermal effect on the flow field. Allegrini et al. (2012a) analyzed the convective heat transfer at building façades in several urban configurations, using the adaptive wall function approach developed by Defraeye et al. (2011) and Allegrini et al. (2012b); they concluded that the AWF provides more accurate heat transfer analysis in urban CFD studies. Offerle et al. (2007) used wind and temperature measurements to examine the thermal structure within a street canyon. They found that buoyancy effects were not seen to have as large an impact on the measured flow field as has been shown in the numerical experiments. Kovar-Panskus et al. (2002) performed a

* Corresponding author. Tel.: +39 06 44 58 56 64; fax: +39 06 48 80 120.
E-mail address: simone.bottillo@uniroma1.it (S. Bottillo).

0038-092X/$ - see front matter
http://dx.doi.org/10.1016/j.solener.2013.10.031

wind tunnel study of the influence of wall-heating on the flow regime in a simulated canyon. They found little evidence of thermal effects, except in a very thin layer near the heated wall, as Louka et al. (2002) noted from an experimental campaign in Nantes, France. Most of numerical studies have been performed on infinitely long street canyons and the prevailing wind direction has been assumed perpendicular to them, so that the spatial domain has been simplified from 3-D to 2-D. In this study, simulations have been performed on a 3-D domain, investigating the impacts of solar heating and ambient wind speed on the flow fields and surfaces temperatures in a street canyon with a fixed H/W and L/W ratios. It has been chosen to study a street canyon isolated from the urban environment, as Blocken et al. (2007) and Allegrini et al. (2012a) did, to evaluate how the buildings configuration affects a spatially homogeneous dynamic and thermal field. Using the commercial CFD code Ansys-Fluent, a series of numerical tests were performed to point out the differences between transient and steady simulations and to investigate the effects of thermal field on the surface temperature on the heat transfer coefficient and air velocity and turbulence.

2. CFD numerical model

In this section, the parameters of the computational model and the boundary conditions are outlined.

The simulations have been performed with the commercial CFD code Ansys Fluent 14.0, 3D double precision, pressure based version and the steady RANS equations have been solved in combination with the standard k–ε model. The governing equations can be expressed as follows:

Momentum equation:

$$\overline{u_j}\frac{\partial \overline{u_i}}{\partial x_j} = -\frac{1}{\rho}\frac{\partial \bar{p}}{\partial x_i} + \frac{\mu}{\rho}\frac{\partial^2 \overline{u_i}}{\partial x_i \partial x_j} - \frac{\partial}{\partial x_j}\left(\overline{u_i' u_j'}\right) + f_i. \tag{1}$$

Continuity equation:

$$\frac{\partial \overline{u_i}}{\partial x_i} = 0. \tag{2}$$

Heat conservation equation:

$$\overline{u_i}\frac{\partial \overline{T}}{\partial x_i} + \frac{\partial}{\partial x_i}\left(K_T \frac{\partial \overline{T}}{\partial x_i}\right) = 0, \tag{3}$$

where $\overline{u_i}$ is the average speed of air flow; $\overline{u_i' u_j'}$ is the Reynolds stress; ρ is the air density; μ is the molecular viscosity; f_i is the thermal-induced buoyant force; $\overline{T}$ is the potential temperature; K_T is the heat diffusivity. The standard k–ε model has been used to solve the turbulence problem. The turbulence kinetic energy, k, and its rate of dissipation, ε, are obtained from the following transport equations:

$$\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_i}(\rho k u_i) = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right] + G_k + G_b - \rho\varepsilon \tag{4}$$

and

$$\frac{\partial}{\partial t}(\rho\varepsilon) + \frac{\partial}{\partial x_i}(\rho\varepsilon u_i) = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_\varepsilon}\right)\frac{\partial \varepsilon}{\partial x_j}\right] + C_{1\varepsilon}\frac{\varepsilon}{k}(G_k + C_{3\varepsilon}G_b) - C_{2\varepsilon}\rho\frac{\varepsilon^2}{k} \tag{5}$$

where G_k is the generation of turbulence kinetic energy due to the mean velocity gradients; G_b is the generation of turbulence kinetic energy due to buoyancy; $C_{1\varepsilon}$, $C_{2\varepsilon}$ and $C_{3\varepsilon}$ constants and the K_T and μ_t expressions are reported in the standard k–ε model of Ansys Fluent 14.0, 2011; σ_k and σ_ε are the turbulent Prandtl numbers for k and ε, respectively. To evaluate the impact of thermal effects, the incompressible ideal gas module has been used for air density.

The simulated urban canyon has the following characteristics: it has an aspect ratio $H/W = 1$ and $L/W = 5$, the orientation is N–S, the buildings width and height are 20 m, the street width is 20 m and the street length is 100 m. Based on the best practice guidelines by Franke et al. (2007) and Tominaga et al. (2008), the dimensions of the computational domain have been chosen in relation to the buildings height H (Fig. 1): the distance between the side walls of the buildings and the north, east and south planes is $5H = 100$ m, instead the west plane is $15H = 300$ m from the westerly building. The distance between the roofs of the buildings and the upper plane is $5H = 100$ m. The buildings dimensions determine the different domain extension behind the built area. When the flow direction is transversal to the canyon length, the obstacle size is maximum and the flow re-development requires a distance of $15H$ from the buildings to the outflow bounds. Instead when the flow is longitudinal to the canyon direction, the obstacle size is minimum and the distance behind the built area is $5H$. Those dimensions allow to set the zero static pressure on the outlet plane and zero gradients of all variables at the top and lateral sides of the domain. The domain dimension over the buildings have been chosen to take into account the blockage ratio, defined as the ratio of the area blocked by the buildings to the total cross-section area. This parameter depends on the obstacles size and wind direction: when the building obstacle area is minimum, the blockage ratio assumes the value of 2%, instead when the wind impacts transversally to the canyon direction, it assumes the maximum value of 5.5%. To simulate the soil influence, the computational domain has been extended 5 m below the ground level. The soil has been simulated setting the following parameters: density = 1000 kg/m3; specific heat = 1000 J/kgK; thermal conductivity = 2 W/mK; temperature at -5 m = 288 K; emissivity = 0.9; solar radiation absorptivity (direct visible and infrared) = 0.8. The building walls have: density = 1000 kg/m^3; specific heat = 1000 J/kg K; thermal conductivity = 0.15 W/m K; thickness = 0.30 m; internal air temperature = 299 K; emissivity = 0.9; solar radiation absorptivity (direct visible and near infrared) = 0.8. The resulting domain size is: 17.25×10^6 m^3.