Energy and Buildings 118 (2016) 142–151

Contents lists available at ScienceDirect

Energy and Buildings

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

How temperature affects the airflow around a single-block isolated building

F. Nardecchia*, F. Gugliermetti[1], F. Bisegna[1]

SAPIENZA University of Rome, Department of Astronautical, Electrical and Energy Engineering, Via Eudossiana 18-00184 Rome, Italy

ARTICLE INFO

Article history:
Received 16 February 2016
Received in revised form 29 February 2016
Accepted 1 March 2016
Available online 4 March 2016

Keywords:
Ventilation
Solar radiation
CFD
Urban environment

ABSTRACT

This study examines the influence of outdoor phenomena together with temperature changes around an isolated building. Considering different wind velocities, temperature differences and building heights, several numerical CFD simulations were performed to investigate the effect of the temperature on the recirculation zones, for various characteristics of the ambient flow field. In the first part, the temperature effects between undisturbed air and building surface were analyzed together with their impact on the flow field. In the second part it was also taken into consideration the influence of the solar radiation. Finally the study was completed by evaluating how different building geometries can affect the previous scenarios.

The results show the importance of the temperature variation for the ventilation around the building and how it modifies, in a significant way, the fluid dynamics close to the building envelope. The correlations found allow to make a proper estimation of the extent of the recirculation regions near the building.

1. Introduction

Natural ventilation can give an important contribution to the environmental sustainability and energy efficient buildings. This is the reason why, recently, scientific researchers tried to evaluate outdoor and indoor ventilation around buildings. Natural ventilation can be caused by: wind induced by pressure, pressure differences due to temperature changes, or a combination of both. Linden [1], examining two kinds of ventilation, that is mixing and displacement ventilations, determined the rules driving them and established how air moves within a building. Hunt and Linden [2] and Heiselberg et al. [3] studied transient draining flows in a space containing buoyant fluid. Larsen et al. [4] investigated deeply the effect of the ventilation in a room with an opening and Karava et al. [5,6] reviewed the current literature about discharge coefficients of natural ventilation openings.

Ventilation performances can be evaluated through experimental techniques, analytic and semi-empirical formulas, simulations with zonal network models and multizone models, and numerical tools, as Computational Fluid Dynamics (CFD) models.

Experimental techniques are experiments performed in wind tunnels. Through this method Murakami et al. [7], Kato et al. [8] and Ji et. al [9] studied wind components, while Etheridge and Nolan [10] and Jiang et al. [11] examined turbulence models in small confined environments. Karava et al. [12–14] also analyzed the air movement in this kind of environment, while Kato et al. [15] and Bu et al. [16] applied their studies on big area studies.

Etheridge et al. [17–19] adopted analytic studies to perform simple or complex parametric analysis, as the one where he suggested to use dimensionless graphs for the building envelope design, in order to increase natural ventilation performances. Còstola et al. [20] recommended a method for the study of air infiltration and ventilation using the pressure coefficients as input parameters. As previously said, simulations with zonal network and multizone models are a different way to examine this phenomenon, as Còstola et al. [21] for the network model, or Li et al. [22] for single-zone and multizone buildings, demonstrated the efficacy of these models in the presence of multiple openings. Then Hensen [23] investigated the combination of network modeling and thermal building models.

However, despite all these works, Chen [24], while analyzing the devices used to predict the ventilation performance around

* Corresponding author. Fax: +39 064880120
E-mail addresses: fabio.nardecchia@uniroma1.i (F. Nardecchia), franco.gugliermetti@uniroma1.it (F. Gugliermetti), fabio.bisegna@uniroma1.it (F. Bisegna).
[1] Fax: +39 064880120.

http://dx.doi.org/10.1016/j.enbuild.2016.03.003
0378-7788/

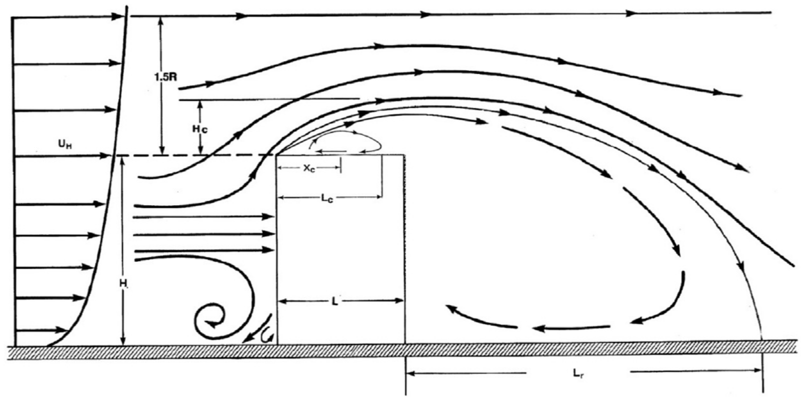

Fig. 1. Pattern of a flow around a rectangular building [42].

buildings, found that analytic and empirical models did not enrich the research literature of the past years, since they did not provide new ideas or solutions to the study of the phenomena afore mentioned. For this reason the numerical tool, in particular the CFD model, became an essential technique used for the ventilation analysis [25]. Many studies were carried out through the CFD to analyze the ventilation in all of its aspects [26–30]. Norton et al. [31,32] carried out a meticulous investigation on both the use of the CFD in general and specific cases of ventilation (agriculture ventilation systems), while Reichrath et al. [33] studied the effects of the ventilation in greenhouses. The numerical approach through the CFD presents some advantages if compared to experimental and analytic techniques, as showed by many papers [23,32,34–36].

Some particular cases, concerning ventilation phenomena around the building, reported by the previous literature were not accurately investigated. For example, there are few studies about how the temperature fields, due to uncommon weather conditions [37], affect the solar radiation [38] or urban apparatus [39,40]. For this reason the goal of this work is to analyze how outdoor phenomena together with temperature changes influence deeply the fluid dynamics around the building [41].

In the present paper a single-block generic isolated building characterized by a wind flow and subject to some particular climatic conditions was studied through the CFD. The study was carried out by varying the following conditions: undisturbed wind velocity, temperature difference between undisturbed air and building, presence and intensity of the solar radiation and building height. The results obtained were then compared with the well-known analytic formulas reported by the literature with particular focus on the AHSRAE Handbook [42] to analyze both visually and quantitatively the ventilation phenomena near the building.

This study concerns only the RANS (Reynolds Averaged Navier-Stokes) turbulence model. Even though the LES (Large Eddy Simulation) is more accurate than RANS model, the latter is still the one most commonly used as in Ratnam and Vengadesan [43] and Vardoulakis et al. notice [44]. For this reason the k-ε model was chosen as the turbulence model, even if it presents some limits for what concerns its performances if compared to other LES model, but on the other hand it is without any doubt more expensive in terms of computation time and resources.

2. Physics problem

One of the most studied cases by the field of wind engineering is the turbulent flow in a 3D cube channel [45,46]. When the flow is near the building, it divides into four main streams, one diverted over the covering, one along the windward façade and the other two deviates to the two sides of the building. There is a stagnation point where the maximum pressure is located. From that point the flow changes its direction, going: toward the areas of the façade where the pressure is lower, upwards, sideways and downwards.

The downward flows forms a vortex in front of the windward façade. Corner streams and areas of flow separation are formed on the sides and top of the construction due to the formation of three vortices, then corner streams converge in a general flow around the corners. Between the vortices and corner streams there is an area with a high speed gradient which is called shear layer and it is located on the buildings corners where the flow separation occurs. Moreover, [46], it was demonstrated that the most visible characteristic of the flow is the formation of a horseshoe shaped vortex around the cube, caused by the separation, on both sides, of the obstacle. The streams at first are separate and then meet again at the downstream of the obstacle.

These phenomena and their geometrical extensions are reported in Fig. 1 [42]:

The goal of this work is to study the change of this distribution when there are temperature gradients and geometrical variations. Particular attention will be given to the extension, in height and width, of the vortex phenomena above the building, through a comparison with the theoretical analysis described in the ASHRAE [42] which defines and describes the zones where these phenomena occur, as follows:

$$H_c = 0.22R \quad (1)$$

$$X_c = 0.5R \quad (2)$$

$$L_c = 0.9R \quad (3)$$

$$L_r = 1.0R \quad (4)$$

where $R = B_S{}^{0.67} B_L{}^{0.33}$ and B_S is the smallest and B_L the largest between the building dimensions in height (H) and depth (W).

L_r is a parameter which is deeply affected by the type of the turbulence model used for the ventilation study, hence it will not be further investigated in this study.

3. Numerical model

The CFD simulations were performed through the commercial software Ansys Fluent v.14.5 [47] using: 3D double precision solver, pressure based solver, steady-state analysis and RANS steady equations were solved in combination with the standard k-ε model. The PISO algorithm was used for pressure-velocity coupling, the pressure interpolation scheme was chosen as second order and second-order discretization schemes were used for the convection and viscous terms of the governing equations. It was assumed the convergence was determined once all the scaled residuals leveled off and reached a minimum of 10^{-6} for x, y and z momentum, k, ε