Energy and Buildings 43 (2011) 3446–3456

Contents lists available at SciVerse ScienceDirect

Energy and Buildings

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

Development and experimental validation of a simulation model for open joint ventilated façades

Cristina Sanjuan[a,*], María José Suárez[b], Eduardo Blanco[b], María del Rosario Heras[a]

[a] Department of Energy, Energy Efficiency in Buildings Unit, CIEMAT, 28040 Madrid, Spain
[b] Universidad de Oviedo, EDZE (Energía), Campus de Viesques, 33271 Gijón, Asturias, Spain

ARTICLE INFO

Article history:
Received 19 July 2011
Received in revised form 22 August 2011
Accepted 5 September 2011

Keywords:
Ventilated façades
CFD validation
Turbulence models
PIV techniques

ABSTRACT

The investigation of the thermal and fluid dynamical behaviour of open joint ventilated façades is a challenging task due to the complex airflows generated inside of the naturally ventilated cavity by the existence of open joints. For this reason, the use of advanced fluid measurement and simulation techniques is highly recommended. This paper focuses in the development and experimental validation of a simulation model for these façade systems. More specifically, different turbulence and radiation models available in the commercial computational fluid dynamic codes have been tested on a three-dimensional model and the results have been compared to particle image velocimetry measurements. The correlation between experimental and numerical data has been used in order to select the simulation procedure for this type of façades. Best fittings have been found when using the RNG k-epsilon turbulence model and the Discrete Ordinate radiation model. Using the selected scheme, parametrical simulations have been performed to investigate the effect of increasing the cavity height, and correspondingly, the number of slabs. Results show that ventilation air flow inside the cavity is enhanced by incident radiation as well as by the height of the façade.

1. Introduction

Generalising about the fluid and thermal performance of open joint ventilated façades (OJVF) is somehow difficult because of the big range of constructive solutions existing in the market. Slabs can be metallic, ceramic or made from stone. Additionally, the dimensions and proportions of the slabs, the shape and size of the open joints, as well as the metallic structure frame supporting the exterior coating differ from manufacturer to manufacturer. Apart from the difficulties derived from the constructive solutions, the existence of open joints distributed along the exterior coating has a great influence in the fluid and thermal behaviour of this façade system in comparison to other continuous ventilated façades, such as double glazed ventilated façades (DGVF), whose behaviour is rather well known, as detailed in the studies made by Manz [1], Safer et al. [2], Baldinelli [3], Fuliotto et al. [4] and Coussirat et al. [5] among other authors. The existence of open joints enables the outdoor air to freely enter and leave the ventilated cavity all along the wall, producing discontinuities and instabilities in the flow inside the ventilated cavity, which is highly dependent not only in the façade geometry by also on the solar incident radiation, outdoor temperature and wind conditions. All these factors, summed to the general lack of data related to these construction systems and the absence of validated models, evidence that there is still a lot of work to do before having a global criterion to determine the energy behaviour of OJVF.

In fact, most technical studies concerning open joint ventilated façades have confined themselves to construction solutions and examining the properties of the materials used in them. The current building standards consider this façades, by default, as ordinary ventilated or slightly ventilated air chamber façades without taking into account their fluid behaviour (CTE [6]). Moreover, the commercial building energy simulation software such as VisualDOE [7], TRNSYS [8] or Energy Plus [9] has not yet included a general purpose model to simulate these façade systems. As a consequence, its actual fluid dynamic behaviour and its performance in terms of energy saving and comfort conditions is not yet sufficiently known. For these reasons, an extensive research of the thermal and fluid behaviour of open joint ventilated façades is thus required in order to turn into reality the claimed advantages (their ability to reduce cooling thermal loads) of these construction systems.

In the last years, some studies related to bottom and top ventilated façades with an opaque outer layer have been published. For example, Griffith [10] proposed a model that was later adopted by the Energy Plus simulation package [9]. It is also significant the computational fluid dynamic (CFD) modelling of a 2D ventilated ceramic façade carried out by Mesado et al. [11], and the work of Patania et al. [12] which is a study of the fluid and thermal

* Corresponding author.
E-mail address: cristina.sanjuan@ciemat.es (C. Sanjuan).

0378-7788/$ – see front matter © 2011 Elsevier B.V. All rights reserved.
doi:10.1016/j.enbuild.2011.09.005

energy performance of three façade types under different temperature and radiation conditions, reporting energy saving rates for the summer period. However, all these models do not consider open joints between the slabs. In their previous works, Sanjuan et al. [13] and González et al. [14,15] investigated the thermal and fluid dynamic phenomena taking place in open joint ventilated façades under solar radiation. They simulated a 2.4 m high ventilated façade composed of four slabs and the corresponding five joints, and compared it to a conventional façade with sealed cavity. Temperatures, velocity profiles and heat fluxes transferred to the room were analysed and discussed. The authors concluded that temperatures in OJVF under radiation conditions are lower than temperatures in the conventional façades with sealed cavity, which means less heat is transferred to the room. The authors also provided a methodology to quantify the energy savings produced by OJVF and compared the results with simplified models of a ventilated cavity used in energy simulation software of buildings. Following the same methodology, Millar et al. [16] simulated also a 2.4 m high ventilated façade composed of nine slabs and ten horizontal joints, and compared its thermal and fluid behaviour to a top and bottom ventilated façade with opaque outer layer. They have obtained significant conclusions although these models have not been validated.

During 2010, Marinosci et al. [17] investigated experimentally and numerically the thermal behaviour of a real ventilated façade. The authors performed temperature, radiation, and velocity measurements in a ventilated façade of a test building with a squared base of 2.89 m and a total height of 7.75 m. The modelling of the façade was made using the software ESP-r and three different air flow nodal network models were tested. The differences of the models depended on whether the joints along the façade were considered sealed or opened. Good agreement between experimental and numerical data was shown when considering the open joints in the modelling. However, very little information on the fluid behaviour of the air inside the ventilated cavity can be extracted from this work.

Also recently, Sanjuan et al. [18] investigated experimentally the fluid and thermal behaviour of an OJVF model in conditions of calm weather and solar radiation. The experimental set up was composed of four slabs and five horizontal joints with a reduced height of 0.825 m. Velocity and temperature measurements were carried out for different heating (solar radiation) conditions. The velocity vector fields were measured with a particle image velocimetry (PIV) system at the vertical centerplane of the cavity. The temperatures in the centre of the slabs and inside the air cavity were measured with Pt100 probes. Additionally, infrared imaging of the slabs surface was also performed. The main features of the flow inside the ventilated cavity are described in the cited reference. The resulting vector fields showed good agreement with previous numerical studies and, in general terms it was observed that the heating of the slabs produces an ascending, but inhomogeneous ventilation flow inside the cavity, that enters though the lower joints and leaves though the upper joints.

The description of the heat and mass transfer phenomena given in the previous works confirms the fact that the complexity of the fluid flow in the regions near the joints and along the ventilated cavity defies analytical methods, making compulsory the use of advanced fluid dynamic simulation techniques, at least in the first stages of the investigation, until all the phenomena taking place are sufficiently known. Nevertheless, the main limitation of the previous cited works relies in the fact that the analysis was performed over specific geometries, making it impossible to expand the results and give a general criterion on the performance of OJVF. Moreover, the application of advanced experimental techniques such as PIV is very laborious, expensive, and restricted to controlled laboratory experiments. For this reason it is very important that numerical models are validated in order to extend its applicability. Once validated, a simulation model can be used in a big range of OJVF geometries and outdoor conditions without the necessity of performing costly experimental measurements.

In a CFD simulation there are some factors that can be intrinsically tested, as the grid, differencing scheme or convergence. Other mathematical tools are more difficult to check or they even depend on the test characteristics. In the OJVF case, these are mainly the procedures employed to simulate the turbulence and radiation. The objective of this work is thus to investigate the suitability of the different turbulence and radiation models available in the commercial CFD codes, to simulate the fluid and thermal performance of OJVF under radiation conditions. The numerical results are compared to PIV measurements performed by Sanjuan et al. [18]. To allow the comparison with these experimental results, the simulations have been performed in a three-dimensional model with the same dimensions of the experimental setup used for PIV measurements. The correlation between the experimental and the numerical data has been used to select the best radiation and turbulence models for the simulation of this façade typology. Main turbulent structures of the flow inside the cavity have been identified and compared. Velocity contours inside the cavity, velocity profiles as well as temperature levels have been used to evaluate the suitability of the mathematical models, and most important, to validate a simulation procedure that could be used with relative confidence, to analyse in detail the thermal behaviour of OJVF systems.

Additionally, the selected method has been applied to analyse the effect of the height of the ventilated cavity and the effects of the incident solar radiation in the convective ventilation flow. The parametrical study aims to provide some valuable information (mass flow though the joints, ventilation mass flow and pressure profiles) that could serve as benchmarking data to validate other models such as those already developed to calculate double glazed ventilated façades (DGVF): models based on energy balances (analytical [19], adimensional [20] or lumped models [21]), models based on nodal airflow networks [22], or models based in control volume discretization [23,24].

2. Experimental set up

For this work, the experimental results have been extracted from a series of experiments performed on an open-joint ventilated façade model. The velocity vectors were measured in the vertical midsection of a laboratory OJVF model. Fig. 1 shows a sketch of the experimental set up. Experiments for three temperature conditions corresponding to Rayleigh numbers $Ra = 5.92 \times 10^8$, $Ra = 9.19 \times 10^8$, and $Ra = 1.35 \times 10^9$ were performed. A summary of the experimental work is presented in the following paragraphs. However, more details on the experimental unit, measurement procedures as well as the experiments conditions and uncertainty calculations can be found in Sanjuan et al. [18].

The PIV system used in the experiments (manufactured by the company TSI) consists of a double cavity Nd:YAG (YAG120-BSL) Laser Pulse Synchronizer, two CCD cameras (630159 Power View 4MPlus) with 4 pixel resolution and dynamic range (12 bit output). The cameras are connected to a PC equipped with a 64 bit frame grabber that acquires 16 frames per second. The Insight 3G Software has been used to control the system. The thermographs of the heated slabs have been obtained with the ThermaCam TM FLIR SC660 also from TSI.

The OJVF model is composed of four slabs made of iron and separated by horizontal open joints. The size of each tile is 30 cm wide, 20 cm high and 2 mm thick. The horizontal joints are 5 mm wide. The air cavity has a thickness of 4 cm. To produce the heating of the slabs (simulating the incident radiation), self-adhesive electrical heating mats were added to each of the four slabs. Surface temperatures were measured in the centre of each slab, and in the middle