Applied Thermal Engineering 37 (2012) 267–274

Contents lists available at SciVerse ScienceDirect

Applied Thermal Engineering

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

Evaluation of various CFD modelling strategies in predicting airflow and temperature in a naturally ventilated double skin façade

Wilmer Pasut [a,*], Michele De Carli [b]

[a] University of California at Berkeley, Center for the Built Environment 390 Wurster Hall #1839, Berkeley, CA 94720-1839, USA
[b] Dipartimento di Fisica Tecnica, Univesità degli Studi di Padova, Via Venezia 1, 35131 Padova, Italy

ARTICLE INFO

Article history:
Received 8 December 2009
Accepted 14 November 2011
Available online 22 November 2011

Keywords:
Double skin façade
CFD
Natural ventilation
Performance simulation
Thermal performance
Air flow

ABSTRACT

Demands for energy savings, thermal and visual comfort and a high-tech image for new building envelopes can be met with a Double Skin Façade (DSF). These kinds of building envelopes are widely encouraged, proposed and increasingly designed by architects. Naturally ventilated DSFs seem very interesting from an energy point of view, but a good design is crucial to improve the energy savings and the proper operation of the system. Computational Fluid Dynamics (CFD) can play an important role in evaluating and improving the thermal behaviour of a DSF. This paper shows, through a sensitivity analysis, a good strategy for carrying out a CFD simulation of this special building envelope. In this work the validations of the results are based on experimental data from the literature.

The paper provides a discussion that highlights which factors are important in the simulation, and which increase model complexity without improving the prediction capacity. The results show that, for a DSF characterized by a prevalent bidirectional flow, the additional effort required to make a 3D model is not justified by a significant improvement of the results. This work shows also that the accuracy can be improved by modelling outdoor ambient. The performance of k-ε and k-ω, the two most commonly used turbulent models for simulating the naturally ventilated DSF is evaluated.

1. Introduction

In the last years, new building envelope systems have been developed in order to improve thermal insulation, to shade solar radiation and to provide suitable thermal and visual comfort conditions. One of these special types of envelopes is “Double Skin Façade” (DSF). DSF are made with two layers of glass separated by a significant amount of air space. The space between the glasses can be ventilated with three different strategies: mechanical ventilation, natural ventilation or hybrid. The ventilation of the air gap contributes to saving energy both during the summer and the winter time. In fact, during the winter time, the air between the glass is heated by the sun rays (greenhouse effect [1]), thus improving the thermal performance of the façade with a consequent reduction of heating costs. With hybrid ventilation systems, during the winter, the fresh air can be pre-heated in the DSF gap before entering in the HVAC system. During the summer, the air flow through the DSF (mechanical or natural) can help to decrease the temperature in the gap.

A blind for solar control is usually installed in the DSF gap. In addiction to reducing heat gain during the summer, this blind increases airflow through the gap with a strong buoyancy effect. In mild seasons, stack effect occurring in the intermediate space can be used as driving force to promote natural ventilation of the whole building [2].

The correct behaviour of a DSF is the key to increasing energy savings, but correct behaviour requires the structure to be designed correctly. One of the weakest spots of this kind of envelope is the design, especially for naturally ventilated façades, where the thermal process and the airflow mechanism influence each other. The magnitude and extent of this interaction depend on the geometric features of system, and the thermal and optical properties of various components.

Ventilated facades are already a common feature of architectural competitions in Europe; but there are still relatively few buildings in which they have actually been realized, and there is still too little experience of their behaviour in operation [1,3,4]. For this reason the CFD analysis could be one of the most important tools to predict the behaviour of DSF and help architects make decisions during the design process.

In the literature there are several examples of using CFD to study the behaviour, features and energy consumption of a DSF [5–7].

* Corresponding author. Tel.: +1 510 642 4950; fax: +1 510 643 5571.
E-mail addresses: wpasut@berkeley.edu, wilmer.pasut@gmail.com (W. Pasut).

1359-4311/$ – see front matter © 2011 Elsevier Ltd. All rights reserved.
doi:10.1016/j.applthermaleng.2011.11.028

The advances in computing power and commercial CFD software available to building mechanical engineers make it possible to use this tool [8]. Using CFD does not necessarily ensure accurate results [9] and it requires engineering judgment [8,10]. Thus the steps of validation, verification, and reporting results described by Chen and Srebric [8] are of great importance.

This research discusses the primary parameters that can influence CFD results during a modelization of natural ventilated DSF. This was carried out through an accurate sensitivity analysis.

The model was compared by using Mei' measurements [11]. These measures were used for two reasons:

- they were carried out in a laboratory, so they were not influenced by wind. The instability of wind can strongly influence the DSF behaviour and make the comparison between CFD and experimental results problematic;
- the velocity and temperature fields inside the gap are presented in the Mei' paper. These can be compared with the CFD velocity and thermal fields to better understand the impact of a different user's choices;

The key points of a sensitive analysis are:

- air property definitions, as constants or as a function of temperature;
- turbulence model;
- presence or absence of external environment;
- 2D or 3D model.

The scope of this work is to show the effects that principal simulation parameters have on CFD results. The scope was not to validate the model. This is because of the other actions, like modifying the CFD model dimensions [5], were not performed in order to improve the results agreement.

The model was realized with the commercial software Fluent [15]. Fluent was used in other similar works [10] [6].

2. Case description

In this work a typical single-story commercial façade was modelized. The main dimensions of a double skin were drawn from an article by Mei [11]. The CFD model was realized with the following dimensions: the outer skin of the façade is a single 12 mm thick clear glass pane, which is 144 cm wide and 206 cm high comprising an aluminium frame. The glass area is 128 cm wide and 191 cm high. In the Mei' case study both the air intake and exhaust of the DSF are designed as a commercial grille arrangement to permit air flow through the façade cavity. The grilles are 24 cm high and 145 cm wide. Each grille has three 4.5 cm high spaces for air ingress and egress. The inner skin is 138 cm $\times$ 200 cm and the glass area is 122 cm $\times$ 185 cm. The sun-shading blind is a venetian type blind. The blind is made of aluminium and is 2.1 m high and 1.45 mwide. The blades are 8 cm wide and the blade angle is 45°. The cavity of air is 55 cm wide and the blind is located at one third of the cavity width as measured from the outer skin [11]. With these dimensions the CFD model was realised. The Mei' test was carried out with fixed values for radiance, inner temperature, and outer temperature. The values for those three parameters are 715 Wm^{-2} for the irradiance (used as a benchmark) and 20 °C for both "indoor" and "outdoor" temperature.

3. Boundary conditions and numerical methods

The inside air temperature (temperature inside the room) was the same as outside, 293.15 K. The air ingress and egress are modelled as a pressure inlet and pressure outlet with the same gauge total pressure equal to 0. In Fig. 1a simplified section of the double skin is shown.

The solar radiation was not directly simulated but the surface temperatures measured by Mai et al. were used as boundary conditions. This was advantageous considering the next step of this work, coupling of CFD program with the energy simulation program "EnergyPlus" [12]. The method of combining CFD and ES program will be the Virtual Dynamic Coupling (as proposed by Chen and Van Der Kooi [13], and [14]), where temperatures are one of the information exchanged between the two software.

The air density was defined as a polynomial function of temperature as follows,

$$\rho = 3.34697 - 0.01055708 \cdot T + 1.10772e^{-5} \cdot T^2 \qquad (1)$$

For this function the program Refprop 7 has been used.

The simulations used the first-order upwind scheme for all of the variables except pressure. The pressure discretization used the Body-Force Weighted scheme [15]. The SIMPLE algorithm was adopted to couple the pressure and the momentum equations. Usually if the sum of absolute normalized residuals for all of the cells in the domain became less than 10^{-6} for energy and 10^{-3} for other variables, the solution was considered converged [16], but the simulations show that the velocity and temperature fields still changed after convergence. For this reason, to be sure the results were stable, the number of iterations was doubled.

3.1. Mesh features

The mesh and the relative number of cells is a critical parameter that strongly influences the computational time. Increasing the number of cells can often increase computational time by an order of magnitude. Furthermore, the grid dimensions influence the accuracy of CFD results and the value of y^+. The parameter y^+ is critical to the correct use of turbulence models. Before doing the simulations different meshes were tested, for 2D and 3D models, in order to figure out the minimum amount of cells that can guarantee the invariability of the results. An important feature of the mesh is that the y^+ value must be less than, or close to, 1 for the first grid close to the walls. That has permitted the use of k-ε with enhanced wall treatment, and k-ω models as turbulent models.

In the 2D models the mesh used has:

- 436 000 nonuniform cells. The corresponding y^+ was about 0.6 for the first grid close to the walls, and the computational time was 7 s per iteration

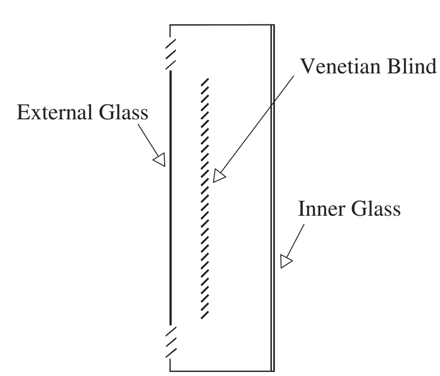

Fig. 1. Section of double skin façade.