ELSEVIER

Available online at www.sciencedirect.com

ScienceDirect

Energy Procedia 78 (2015) 1537 – 1542

Energy Procedia

6th International Building Physics Conference, IBPC 2015

Modelling natural ventilation in double skin facade

Diego Angeli[a], Alessanro Dama[b]*

[a]Department of Sciences and Methods for Engineering – University of Modena and Reggio Emilia, Via Amendola 2, 42122 Reggio Emilia, Italy
[b]Department of Energy - Politecnico di Milano,Via Lambruschini 4, 20156 Milano, Italy

Abstract

The assessment of the energy performance of buildings with Double Skin Facades (DSF) requires proper dynamic simulation tools, based on models capable of predicting heat and mass transfer in the DSF under variable boundary conditions, at the price of a reasonable computational effort.

Many DSF simplified models have been developed and implemented in building simulation tools, but the validation of these tools is still an open issue, especially for the prediction of the mass flow rate in naturally ventilated DSF. The CFD modelling activity presented in this work aims at investigating the reliability of the assumptions and hypotheses employed in the simplified model, which was specifically developed for the dynamic simulation of heat transfer in buildings.

Both the CFD and simplified models have been tested and evaluated on an experimental case study, using the database provided by a research program developed under IEA ECBCS Annex 43/SHC Task 34, reporting the results of a measurement campaign conducted on an a transparent naturally ventilated DSF tested in Denmark, in an experimental facility called “the Cube”.

Peer-review under responsibility of the CENTRO CONGRESSI INTERNAZIONALE SRL

Keywords: natural ventilation; CFD; double skyn façade; model validation.

1. Introduction

The assessment of the energy performance of buildings with Double Skin Facades requires proper dynamic simulation tools, based on models capable of predicting DSF heat transfer under variable boundary conditions, at the price of a reasonable computational effort.

---

* Corresponding author. Tel.: +39 02 2399 3864; fax: +39 02 2399 3868.
E-mail address: alessandro.dama@polimi.it

Peer-review under responsibility of the CENTRO CONGRESSI INTERNAZIONALE SRL
doi:10.1016/j.egypro.2015.11.186

Despite in the last 10-20 years many authors have faced the issue of modeling DSFs, by means of bulk temperatures, thermal networks, lumped parameters, single and multi-zone approaches, the validation of simulation tools is still an open issue, especially for naturally ventilated DSFs. Indeed within a recent research program developed under IEA ECBCS Annex 43/SHC Task 34 [1] a monitoring campaign on a DSF case study has been carried out in order to provide data for the validation of dynamic simulation tools [2]. It has been shown that the performances of present building simulations tools against measurements are not yet satisfactory especially for the prediction of the mass flow rate in naturally ventilated DSFs. [3].

The CFD modelling activity presented in this work aims at investigating the reliability of the assumptions and hypotheses employed in the simplified model, which was specifically developed for the dynamic simulation of heat transfer in buildings. The simplified model, indeed, is based on a pressure loop and on an integral approach to the heat transfer. It considers the buoyancy as a function of the average temperature in the vertical channel. Uniform heat transfer coefficients, depending on the geometry and flow regime, are adopted in order to represent convection inside the channel, according to the available correlations for wide or narrow channels.

Both the CFD and simplified models have been tested and evaluated on an experimental case study, using the database provided by a research program developed under IEA ECBCS Annex 43/SHC Task 34, which reports the results of a measurement campaign conducted on an a transparent naturally ventilated DSF tested in Denmark, in an experimental facility called “the Cube”.

2. Methods

2.1. Case study

For the assessment and the validation of both the Simplified and the CFD model was used the data set of an experimental campaign on a naturally ventilated DSF performed by the Department of Civil Engineering of the Aalborg University, Denmark, within the IEA ECBCS Annex 43/SHC Task 34 [1]. The test facility, named “The Cube”, is located in a suburban area, with a double skin façade 3,555 m X 5,450 m facing South. A detailed description of the facility can be found in [2].

The measurement campaign refers to the period 1-15 October 2006. Limiting the present CFD investigation to focus on the buoyancy and surface heat transfer issues, we have selected October the 1st at 2 pm, a day with a considerable solar irradiance and a negligible wind action. The external air temperature was 19.2 °C and in the DSF channel the temperature of the external glass was this are the main input (i.e. boundary condition) for both the CFD and the simplified model. The model results will be compared with the measured mass flow rate and air temperatures in the DSF at different heights.

2.2. Simplified model

The Simplified model is based on the integral formulation already employed in [4] and a pressure loop scheme, respectively devoted to describe the thermal and the fluid-dynamic problem. They are solved together by means of simple iterations on the volume averaged temperature and the averaged air velocity in the channel. An extensive description was given in [5].

The convection inside the channel is modelled employing bulk temperatures and uniform surface heat transfer coefficients. With reference to a 2D representation, the bulk temperature in the channels is defined in equation (1):

$$T(y) \equiv \frac{1}{s}\int_{0}^{s} \rho(x,y)u_y(x,y)T(x,y)dx \tag{1}$$

The equations for the thermal problem are based on the thermal balance adopted by standard EN 13362-2 and reported below (2):

$$\begin{cases} [h_{cv1}(T_{w1} - T(y)) + h_{cv2}(T_{w2} - T(y))]L dy = \dot{m} c_p dT \\ T(0) = T_{inlet} \quad 0 \le y \le H \end{cases} \Rightarrow T(y) = \overline{T}_w + (T_{inlet} - \overline{T}_w)\, e^{-ky} \quad \begin{cases} \overline{T}_w \equiv \dfrac{h_{cv1} T_{w1} + h_{cv2} T_{w2}}{h_{cv1} + h_{cv2}} \\ k \equiv \dfrac{(h_{cv1} + h_{cv2})}{s\, v_m \rho_m c_p} \end{cases} \qquad (2)$$

The surface heat transfer coefficients are derived by correlations in the literature. A specific review for DSF problems is given in [6]. The main issue is the choice between wide or narrow channel correlations. In the former case the velocity and temperature boundary layers from the two walls are almost separated, so that correlations for single plates can be used, while in the latter case the two layers are completely overlapped and correlations for parallel plates should be employed. The wide channel can be identified by the following criteria: $Ra_H^{1/4} > H/s$, where H is the height and s the thickness of the ventilated air gap

As already shown by [7], the inlet air temperature can significantly differ from the outdoor temperature. For this purpose a model correction was developed for this particular case study. It takes into account the preheating effect on inlet air due to the solar radiation entrapped and absorbed at the basement of the DSF cavity.

The pressure loop scheme employs non dimensional discharge coefficient *Cd* at the openings are, evaluated according to ASHRAE standards and non-dimensional wind pressure coefficients *Cp*, evaluated at the opening heights, according to the measured wind velocity and direction and the *Cp generator* results reported in [8].

2.3. CFD model

Two-dimensional, steady-state numerical simulations are also performed in the present work, with the twofold purpose of drawing a comparison with the results of the analytical model described above, and of validating the numerical approach itself in light of the experimental benchmark. The calculations are carried out by means of a finite-volume solver based on the open source OpenFOAM® computational toolbox [9], implementing the Reynolds-Averaged Navier-Stokes and energy conservation equations governing non-isothermal, turbulent flow, as well as several turbulence models [10]. Due to the presence of several significant temperature differences in the problem at hand, two alternative approaches have been adopted for the modeling of density (and other fluid properties): (i) in first instance, the Oberbeck-Boussinesq approximation [11] has been assumed to be valid, all fluid properties being taken as constant except for density in the buoyancy term; (ii) as a counterpart, further calculation have been run by enforcing the ideal gas law in all occurrences of density, all other properties being kept constant.

Different turbulence models have been associated to each of the two alternative models for density; in particular, the q-ζ model by Gibson and Dafa'Alla [12] has been adopted for the calculations under the Boussinesq approximation, whilst the low-Reynolds k-ε model of Launder and Sharma [13] has been chosen for the ideal gas simulations. Such a peculiar choice was dictated by the current availability of turbulence models in OpenFOAM® for incompressible and compressible flow, respectively [14], and by numerical stability issues arisen when applying the Launder-Sharma k-ε model in its incompressible version.

The 2D computational domain for the simulations is schematized in Figure 1(a). The shape of the double-skin façade of the "Cube" has been represented in the best possible detail, based on the available documentation [2], including the outline of the window framing. The domain boundaries have been subdivided as in Figure DOMAIN, and uniform temperature conditions have been imposed at all glazed surfaces of the façade, as well as at its ground and ceiling. Such temperature values are directly taken from the experimental data, as detailed in the following section. Hence, the real physics of radiation and of conjugate heat transfer between the glazing and frames and the surrounding air, has been substituted with the assumption of constant wall temperature. Albeit strong, such an approximation is deemed to be sufficiently realistic for the scope of the present work.

Details of the block-structured computational grid are displayed in Figure 1(b) and (c). The total number of quadrilateral mesh elements amounts to 300000. Near-wall sizing has been chosen so as to avoid the resort to wall functions and, hence, to reproduce the mean features of the diffusive sub-layers. Second-order schemes were adopted for the discretization of both the convective and diffusive terms. The SIMPLE algorithm [15] has been employed for the de-coupling of pressure and velocity in the Boussinesq cases, while a fully coupled solver has been enforced under the ideal gas assumption.

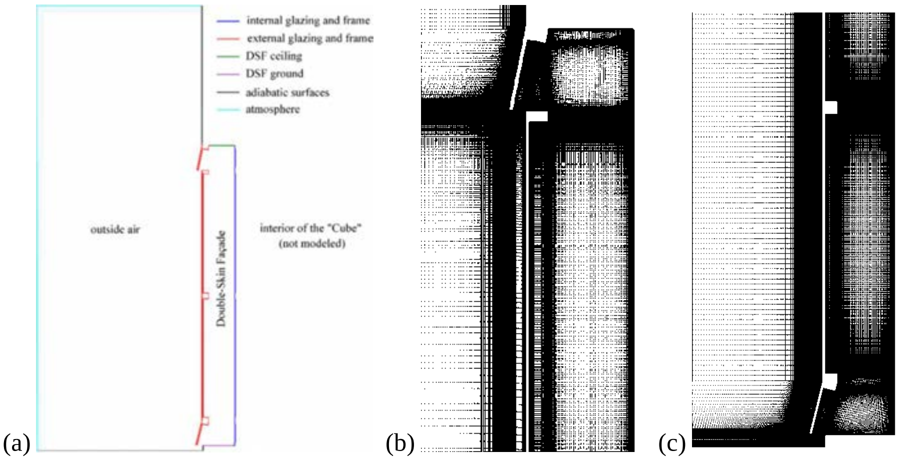

Fig. 1. (a) CFD domain; grid details: DSF top (b) and bottom (c) parts.

3. Results

Table 1 reports the experimental results in terms of mass flow rate per unit width, temperature increase and total heat removed as measured in the DSF, and the relative deviations associated with each of the modeling approaches adopted. It can be observed that both the CFD and simplified models produce reasonable estimates of such integral data, with differences linked to modeling choices. In particular, CFD with the Boussinesq hypothesis and q-ζ model produces a rather accurate estimate of the temperature increase, but strongly underestimates the mass flow rate and, hence, the total heat flux; conversely, with the ideal gas assumption and Launder-Sharma (L-S) k-ε model, the flow rate is captured more accurately, while the temperature difference and the heat flux are both slightly underpredicted.

Table 1. Experimental results and relative deviations of model estimates.

Measurements and model variants	Experimental	CFD id. gas+ L-S k-ε	CFD Bouss.+ q-ζ	SM no preheating	SM preheating	SM preheat & -50% Cd	SM preheat & +50% Cd
T diff. (out. –ext.)	7.1 K	-9.2%	0.7%	-2.0%	-1.8%	-20.6%	10.8%
Mass flow rate	0.0622 kg/s/m	-4.6%	-22.3%	-2.7%	2.2%	28.9%	-10.7%
Heat removed	476 W/m	-13.3%	-21.7%	-4.7%	0.3%	2.4%	-1.1%

The simplified model provides a remarkably accurate estimate of all quantities; however, predictions of the temperature differences and mass flow rates appear to be extremely sensitive to the discharge loss coefficient, while the overall heat flux is always well captured. The substantial concordance between models and experimental data is also confirmed by the trends of bulk temperature plotted in Fig. 2(a). It is to be noted that all models seem to underestimate the bulk temperature profile, as compared with measurements, except for the sole outlet temperature, which is instead captured quite accurately. Such a discrepancy could partially be ascribed to a possible bias affecting temperature measurements in hours of strong irradiation, although it is claimed from the source of the experimental data [2] that suitable shielding had been employed to prevent systematic errors due to direct insolation.

Figure 2(b) reports temperature maps and contours as obtained by CFD, for both the Boussinesq/q-ζ and ideal gas/L-S k-ε cases. It is observed that the general pattern is substantially analogous, its main characteristic being the buoyant jets arising from the inlet and outlet sections of the façade. Nevertheless, the shape of the contours in the ideal gas/L-S k-ε case suggests that turbulent effects are underpredicted with respect to the Boussinesq/q-ζ case. This difference is however likely to be due to the different turbulence model adopted, rather than to the assumptions on density. Such indications are confirmed by the velocity profiles shown in Fig. 3, as sampled at different heights along the DSF, which also denote the constant presence of two separate boundary layers along the external and internal wall, thus validating the large gap assumption made in the simplified model.

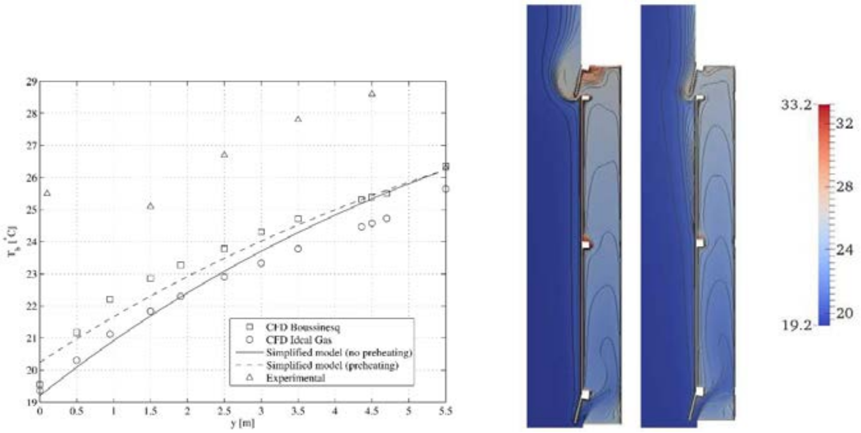

Fig. 2. (a)Vertical profiles of bulk temperature (b) Temperature contours from CFD: (left) Boussinesq/q-ζ, (right) ideal gas/L-S k-ε.

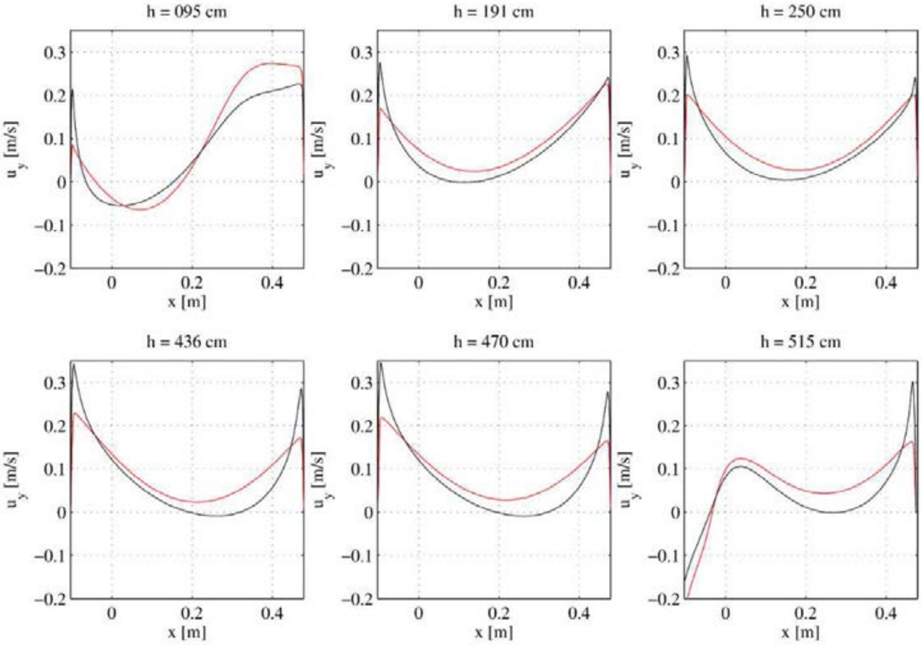

Fig. 3. y-velocity horizontal profiles at given heights from CFD: Boussinesq/q-ζ (black) vs. ideal gas/L-S k-ε (red).

Finally, profiles of the local heat flux along the internal and external glazing surfaces of the DSF, as obtained by the CFD and simplified models, are plotted in Fig. 4. In the CFD results, it can be observed that the impingment of the inflowing buoyant jet augments heat transfer on the internal wall near the bottom of the DSF, and the presence of the frames induce local detachments and reattachments of the boundary layers, thus implying strong gradients. The profiles obtained with the simplified model are obviously more regular, but it is important to note that they show a qualitative agreement with the trends predicted by CFD in the regions where the boundary layer is attached. Such an occurrence corroborates the viability of the assumption, made in the simplified model, of a constant heat flux along the façade walls.

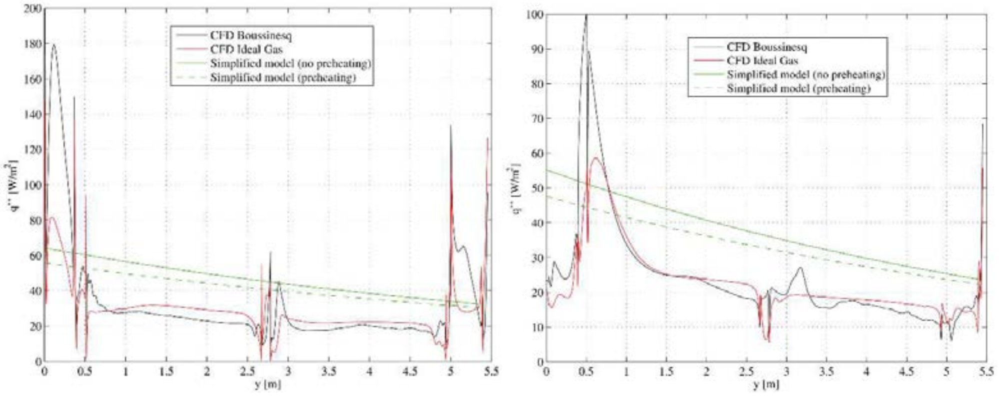

Fig. 4.Heat fluxes: (a) external glass; (b) internal glass.

4. Conclusions

A cross-comparison between a semi-empirical analytical model and CFD has been carried out for the case of a naturally ventilated double-skin façade, for which an extensive experimental dataset is available from literature. The results show that both models produce reasonable estimates and, in some cases, very accurate guesses of both local and global quantities, in spite of a considerable sensitivity to the different modeling approaches. Such a favorable comparison encourages future analysis for further assessment of the methodologies.

Acknowledgements

The authors warmly thank Dr. Olena Kalyanova for kindly providing the experimental data set of “The Cube” test facility, used here for the model validation.

References

[1] IEA ECBCS Annex 43/SHC Task 34, http://www.ecbcs.org/annexes/annex43.htm

[2] Kalyanova O, Heiselberg P. Experimental Set-up and Full-scale Measurements in ‘The Cube'. Aalborg University, Department of Civil Engineering. 2007.Technical Report No.34.

[3] Kalyanova O, Heiselberg P, Felsmann C, et al. An empirical validation of building simulation software for modeling of double skin façade (DSF), Proceedings of 11th International IBPSA Conference, 2009. pp. 1107-1114.

[4] Angelotti A, Dama A, Mazzarella L, Perino M. Validazione sperimentale di un modello per facciate a “doppia pelle” in ventilazione meccanica, Proceedings of 62nd National Congress ATI, 2007.Vol. 1, pp. 306-311 (in italian).

[5] Angelotti A, Dama A, Mazzarella L. Modeling naturally ventilated double skin facades. In: CLIMA 2010 10th REHVA World Congress "Sustainable Energy Use in Buildings". Antalya, Turkey, 9-12/05/2010,

[6] Saelens D. Energy Performance Assessment of Single Storey Multiple Skin Facades, PhD Thesis 2002. Leuven (Belgium).

[7] Saelens D, Roels S, Hens H. The inlet temperature as a boundary condition for multiple-skin façade modeling, Energy and Buildings, 2004.vol. 36, pp. 825-83.

[8] Costola D, Blocken B, Hensen JLM, Overview of pressure coefficient data in building energy simulation and airflow network programs. Building and Environment 2009. Vol. 44, pp. 2027-2036.

[9] Weller HG, Tabor G, Jasak H, Fureby C. A tensorial approach to computational continuum mechanics using object orientated techniques. Comp. Phys. 1998;12:620-631.

[10] Wilcox DC, Turbulence Modeling for CFD. 3rd ed. DCW Industries; 2006.

[11] Gray DD, Giorgini A. The validity of the Boussinesq approximation for liquids and gases. Int. J. Heat Mass Transfer 1976;19:545-551.

[12] Gibson MM, Dafa'Alla AA. Two-equation model for turbulent wall flow. AIAA Journal 1994;33:1514-1518.

[13] Launder, BE, Sharma BI. Application of the Energy-Dissipation Model of Turbulence to the Calculation of Flow Near a Spinning Disc. Letters in Heat and Mass Transfer 1974;1:131.

[14] http://www.openfoam.com

[15] Patankar SV. Numerical Heat Transfer and Fluid Flow. 1st ed. Hemisphere, Washington DC; 1980.