The influence of different flow velocities on the heat transfer inside a ventilated façade*

Monica Chereches[1], Nelu-Cristian Chereches[2#], Sebastian Hudisteanu[2]

[1]National Institute for Research and Development in Construction, Urban Planning and Sustainable Spatial Development, NR&DI URBAN-INCERC Iaşi Branch
37 Anton Sesan street, 700048, Romania
E-mail: putina_monica@yahoo.com

[2]Technical University "Gheorghe Asachi" of Iasi, Faculty of Civil Engineering and Building Services, Department of Building Services Engineering
13, bd Dimitrie Mangeron, 700050, Iasi, Romania
E-mail: chereches@tuiasi.ro, seby_hudisteanu@yahoo.com

Abstract. *In this paper, a comparative numerical study has been conducted in order to analyze the thermodynamic behavior inside a channel of a double skin ventilated façade in cold season with air supply ventilation mode. The study was performed in forced convection with CFD software Ansys-Fluent. A parametric investigation concerning the fluid inlet velocity has been conducted to study its influence on both fluid flow and heat transfer. The obtained results show the temperature and the velocity profiles, for different air velocities at the inlet section in order to calculate the heat loses through the surface of the interior glazing.*

Key words: numerical simulation, forced convection, air velocity, heat flux

1. Introduction

Double skin facades (DSF) are building envelopes composed of two layers of glass separated by a ventilated air channel. Due to the transient and complex air flow in the façade channel, the influence on the indoor environment and energy consumption are very difficult to evaluate. Computational Fluid Dynamics (CFD) can play an important role in evaluating and improving the thermodynamic behavior of a double skin facade. At the moment, there are more than 240 software products for simulations, available for over 20 years. The most known and used of these are: CAPSOL, ANSYS, TRNSYS, ESP-r, TAS, ENERGYPLUS, Ecotect. The performed simulations can be grouped into the following categories [1]: simulations for real buildings and simulation without any connection with these. Among these, Saelens [2] analyzed the energy efficiency of various types of single-storey facades; Balocco [3] used a computer model to study the steady state energy performance of a naturally ventilated façade; Li Hao Yin Chiu et al. [4] investigated the energy savings that

* Lucrare inclusă în programul conferinţei EENVIRO 2013
# correspondent author: chereches@tuiasi.ro

resulting from the preheating ventilation air by means of a double naturally ventilated facades. Kalyanova et al. [5] performed an empirical validation of building models with DSF, with various building simulation tools (ESP-r, IDA ICE 3.0, VA114, TRNSYS-TUD and BSim). In the literature there are also several examples of using CFD to evaluate energy efficiency and thermal performance of naturally or mechanically ventilated facades [6, 7, 8].

It was concluded that none of the models was consistent enough when comparing simulation results with experimental data for the ventilated cavity and only some models showed reasonable agreement with the experimental results for the thermal buffer mode.

Safer [9] realized a comprehensive modelling of a compact double-skin facade equipped with a venetian blind and forced ventilation. It was concluded that the distance between the blind and the external glazing have a major impact on the velocity profiles inside the double-skin facade channel.

In this study a comparative numerical analysis was made for thermodynamic behavior of a double skin façade. The façade channel with 45 cm thickness is composed of a single glazing to the exterior and a double glazing to the interior.

The parametric investigation concerning the fluid inlet velocity has been conducted in order to study its influence on both fluid flow and heat transfer.

2. Case description

The simulations were performed in forced convection for the following hypothesis and boundary conditions:

- channel geometry: 2,8m height, 2 m width and 0,45m channel thickness (fig.1);
- exterior glazing: 2,8m height, 2 m width;
- interior glazing: 1.85m height, 2 m width.

The façade channel is subjected to a uniform solar flux on the external glazing which is refracted to the interior one:

- constant heat flux density of solar radiation on the exterior glazing: $\varphi_e = 56$ W/m^2;
- constant heat flux density of solar radiation on the interior glazing: $\varphi_i = 45$ W/m^2;
- air temperature at inlet section of the channel corresponding to the cold season: $T_e = -18^oC$;
- velocity at inlet section of the channel: v_i = 0,1m/s, 0,3m/s and 0,5m/s (table1);

Table 1

Case studies for different air velocity

Velocity \ Cases	1	2	3
v_i (m/s)	0,1	0,3	0,5

- air supply ventilation mode (ascendant air circulation between exterior inlet and interior outlet sections).

3. Fluid Dynamic Modeling

The analysis was made using CFD software ANSYS-Fluent in turbulent flow using the k-ε RNG turbulence model and the control volume method in order to resolve the continuity, the momentum and the energy equations in the steady state on a three-dimensional model (fig. 2). Moreover, a uniform rectangular network of nodes was performed for fluid domain. The calculation is an iterative one, the chosen convergence criteria are 10^{-6} for the temperature and 10^{-4} for both the pressure and the velocities. The geometry and the mesh for channel were made with Ansys-DesignModeler.

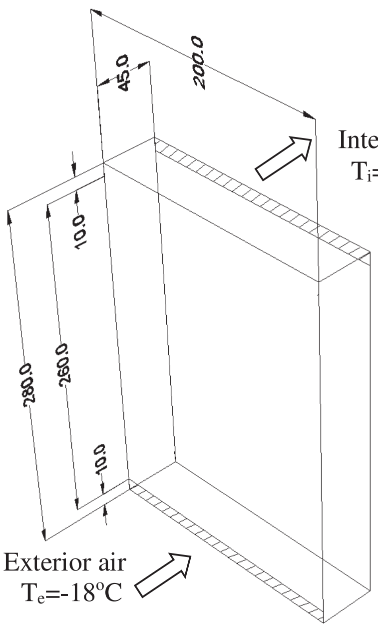

Fig. 1. The 3D configuration of air flow channel

Fig. 2. The meshing of three-dimensional model

The obtained results show the temperature and the velocity profiles inside the channel (fig. 3-5), for different air velocities at the inlet section in order to evaluate the heat loses through the surface of the interior glazing.

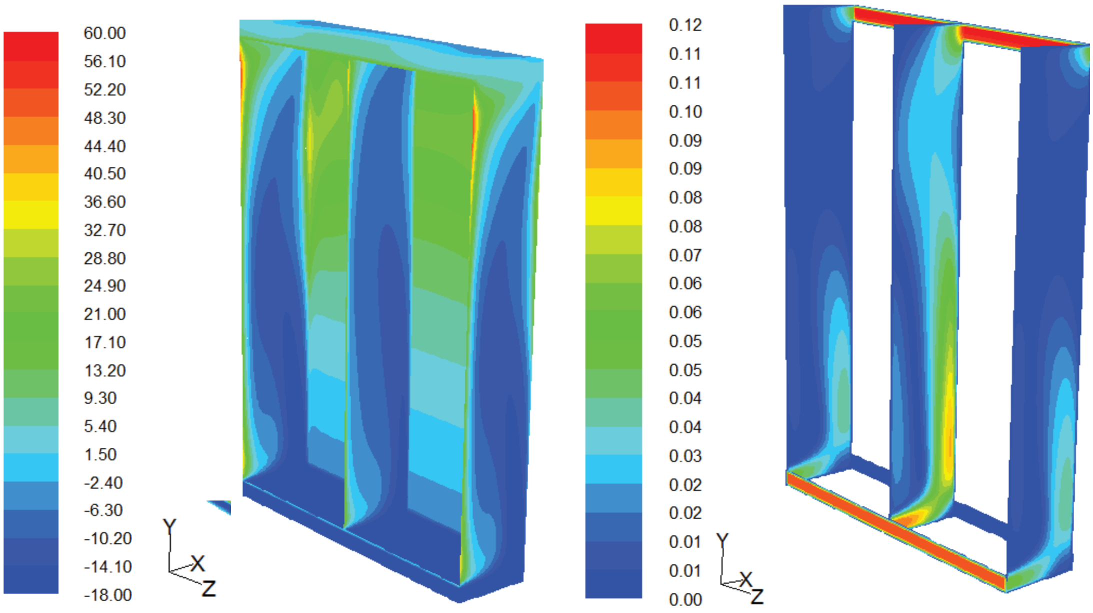

Fig. 3. Temperature and velocity fields inside the channel for case 1

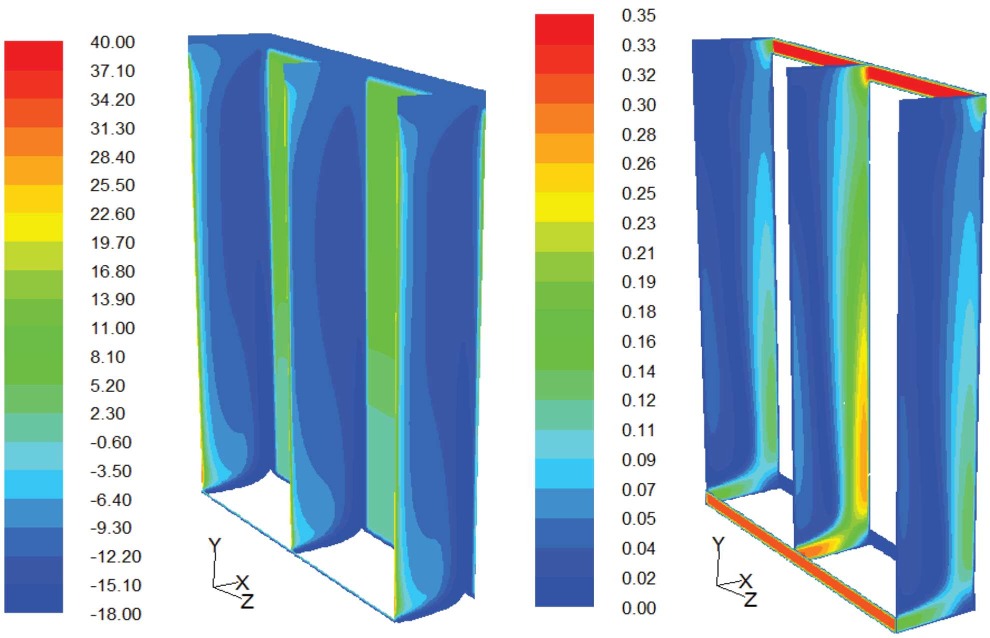

Fig. 4. Temperature and velocity fields inside the channel for case 2

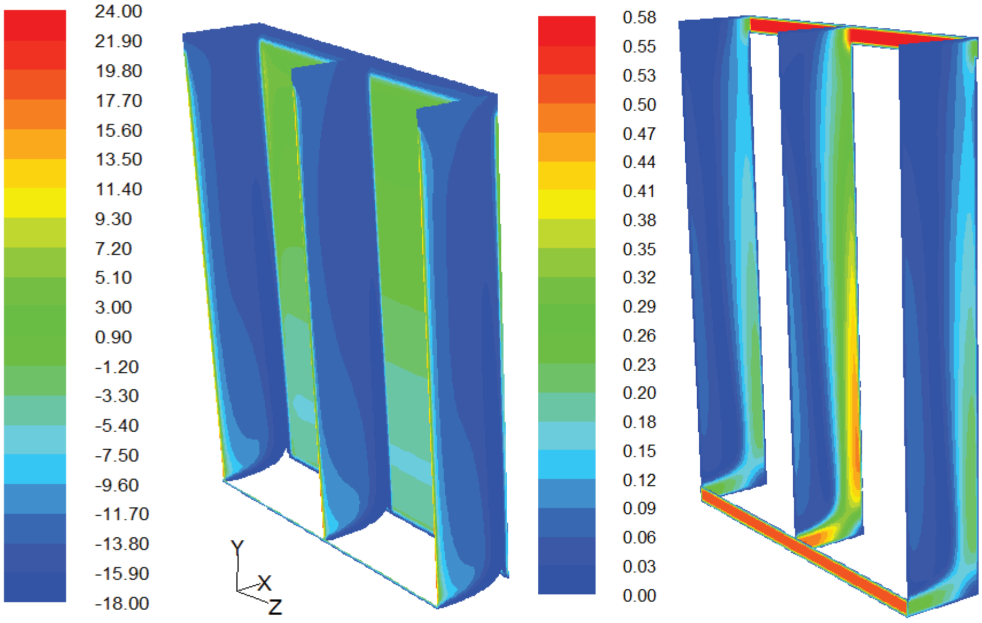

Fig. 5. Temperature and velocity fields inside the channel for case 3

The temperature and velocity profiles in median zone of the channel are presented in fig. 6 - 8 for the three cases.

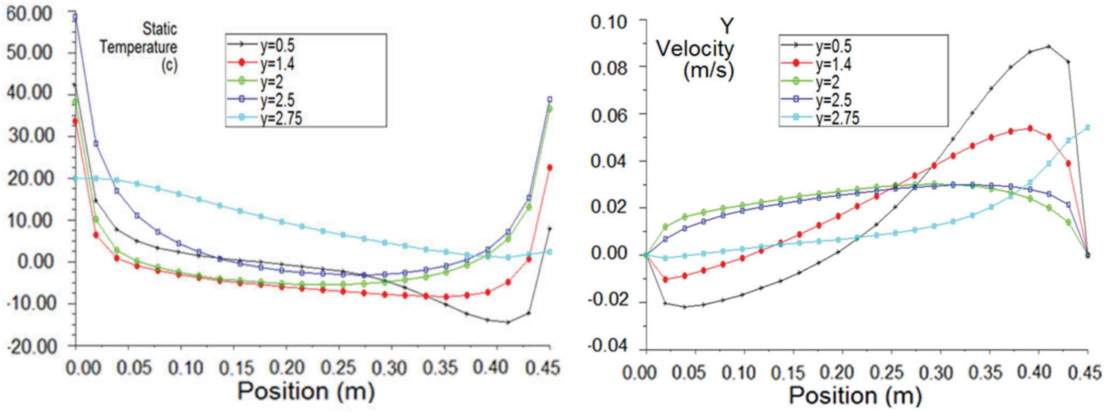

Fig. 6. Temperature and velocity profiles inside the channel for case 1

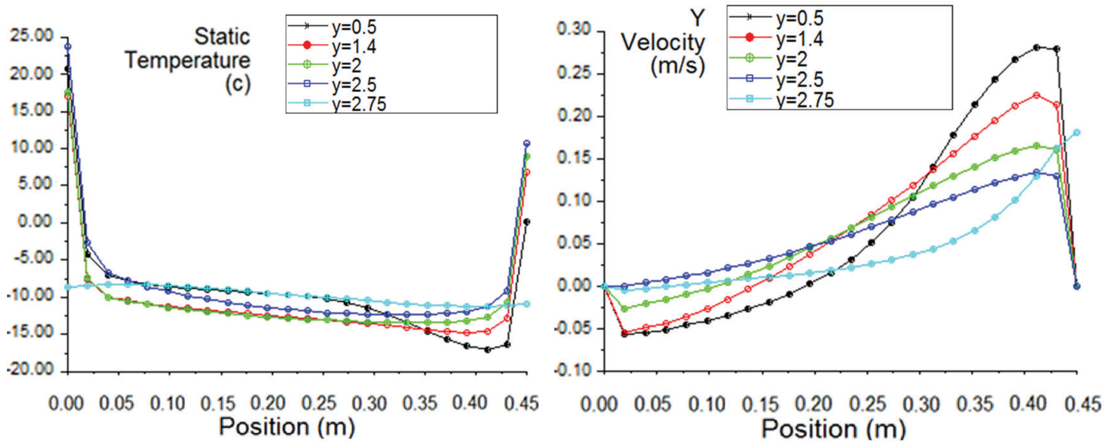

Fig. 7. Temperature and velocity profiles inside the channel for case 2

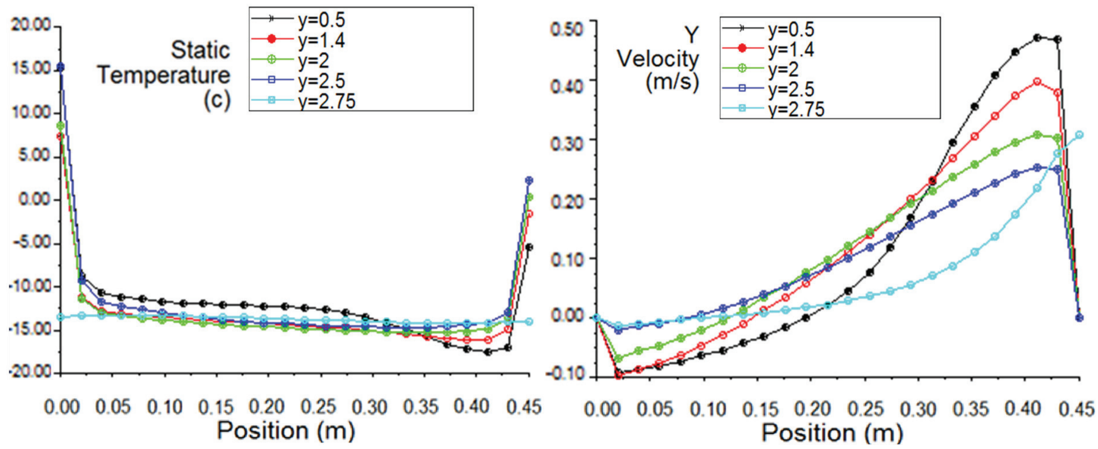

Fig. 8. Temperature and velocity profiles inside the channel for case 3

4. Results and discussion

In table 2, we evaluated the heat flux Q (W) (eq.1) transferred from interior chamber with temperature T_i = 20°C into the double skin channel, through the inner glass with properties: surface S = 3,7 m^2 and thermal resistance R = 0,6 $m^{2o}C/W$ (K=1/R). The temperature of the air channel (T_c) was calculated from numerical simulations as averaged temperature (table 2). Thus, it was observed an increase of heat flux transferred through interior glazing (in terms of recovered heat) with increasing of the air flow velocity (table 2) due to the decrease in air temperature inside the channel.

$$Q = SK(T_i - T_c)\,(\mathrm{W}) \tag{1}$$

Table 2

Heat loses through the surface of the interior glazing

	Case 1	Case 2	Case 3
	v_i = 0,1	v_i = 0,3	v_i = 0,5
T_c (°C)	-1,85	-11,39	-13,80
Q (W)	134,7	193,6	208,4

The numerical results obtained by iteration in ANSYS-Fluent software highlighted the velocities and temperatures profiles and the fluid recirculation phenomena near exterior glass inside the channel which increases with air velocity. (fig. 6-8). Also, it was observed an ascending flow inside the channel near the inner glass surface where velocities are higher values. Consequently, the values of air temperature are higher near the inner glass surface and lower near the exterior glass (fig. 3-5).

5. Conclusions

In this paper, numerical simulation of a double skin façade channel has been conducted in cold season in order to observe the influence of air velocity on the heat transfer through the interior glass surface. After analyzing the results it was found that the flow regime depends on the air velocity inside the channel and the solar flux on glazing. Thus, the spectrum of non-isothermal flows (including the velocity and the temperature) is characterized by the severe impact of the boundary conditions. The numerical simulations reveal that for lower velocities, the air temperature increases inside the channel and the heat loses decreases leading to better energy saving.

Finally, the CFD analyses are important tools to predict the behavior of these facades for the design processes, leading to more simulations in order to take into account different parameters and to propose optimal functional configuration.

Acknowledgement

We gratefully acknowledge the financial support of the Ministry of Regional Development and Public Administration through contract no. 493/2011.

References

[1] *M. Perino et al.*, „State of the Art Review of Responsive Building Elements", Volume 2A, International Energy Agency (IEA), 2008

[2] *D. Saelens, J. Carmeliet, H. Hens*, „Energy performance assessment of multiple skin facades", Int. Journal of HVAC&Research, vol. 9, no. 2, 2003, pp.167-186

[3] *C. Balocco*, „A non-dimensional analysis of a ventilated double façade energy performance", Energy and Buildings, vol. 36, 2004, pp. 35-40

[4] *Y. Chiu, L. Shao*, „A parametric study of solar double skin façade on energy saving effect", Proceedings of the 7th Clima 2001 World Congress, Napoli, Italy, 2001

[5] *O. Kalyanova, P. Heiselberg, C. Felsman, H. Poirazis, P. Strachan*, „An empirical validation of building simulation software for modeling of double skin façade (DSF)", Eleventh International IBPSA Conference Glasgow, Scotland, July 27-30, 2009, pp. 1107 – 1114
[6] *H. Manz, A. Schaelin, H. Simmler*, „Airflow patterns and thermal behavior of mechanically ventilated glass facades", Building and Environment 39, 2004, pp. 1023–1033
[7] *H. Manz*, „Total solar energy transmittance of glass double facades with free convection", Energy and Buildings 36, 2004, pp. 127–136
[8] *A. Zoller, E.R.F. Winter, R. Viskanta*, „Experimental studies of combined heat transfer in turbulent mixed convection fluid flows in double-skin facades", International Journal of Heat and Mass Transfer 45, 2002, pp. 4401–4408
[9] *N. Safer, M. Woloszyn, J.J. Roux*, „Three-dimensional simulation with a CFD tool of the airflow phenomena in single floor double-skin façade equipped with a venetian blind", Solar Energy 79, 2005, pp. 193-203
[10] *D. Constantinescu*, „Contribuţii la calculul şi optimizarea sistemelor pasive şi mixte de încălzire solară", PhD Thesis, ICB, 1988.