Revista Română de Inginerie Civilă, Volumul 6 (2015), Numărul 1

Fluid flow and heat transfer simulations for a box double-skin façade in Brasov, Romania*

Gabriel NASTASE[1], Robert GAVRILIUC[2], Alexandru SERBAN[1], Ioan BOIAN[1],

[1]Civil Engineering Faculty, Transylvania University, str. Turnului nr. 5, Brasov, Romania
[2]Building Services Faculty, Technical University of Civil Engineering, Bucharest, Romania

traznasa@gmail.com

Abstract: *CFD – Computational Fluid Dynamics Method is the modern scientific method, for prediction of fluid flow, heat and mass transfer and other related phenomena, by numerical solving the mathematical equations governing those phenomena, such as the law of mass, momentum and energy conservation. CFD method is very useful when it comes to conceptual studies for new model of buildings, detailed development of certain parts of a building, useful for solving problems that appear during construction or exploitation of a particular system that is part of a building etc. CFD analysis complements the numerical modeling and experimental modeling, reducing the effort and cost for experiment and data acquisition systems. This paper presents a few results for heat transfer and fluid flow, made on a box double-skin façade, in Brasov, Romania, as a part of a PhD thesis.*

Keywords: *heat transfer, fluid flow, box double-skin façade, convection, heating, cooling, energy.*

1. Introduction

This article presents a few results for heat transfer and fluid flow obtained from a box window double-skin façade (DSF) model, made after a real model developed in situ, on the ground floor of the Civil Engineering Faculty in Brasov, part of Transylvania University from Brasov, on the south façade of the building. The real model serve as a basis for all the measurements made with the goal of modeling and then validate processes within the system and ultimately determine the impact of such façade on a building placed in Brasov, Romania. The experimental model is part of a PhD thesis and consists of a so-called ”box window”-type double-skin façade connected with the experimental room by an indoor air curtain; it is manually controlled, having a forced air circulation in summer. The venetian blinds shading system is placed inside the cavity, in the median area (see Figure 1).

* Lucrare inclusă în programul conferinţei RCEPB 2014

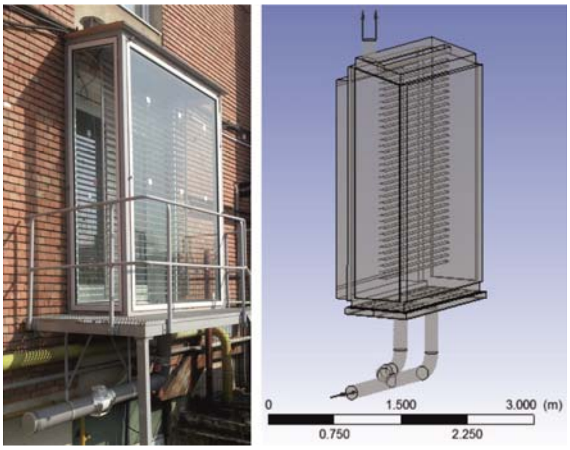


Fig. 1. Box double-skin façade experimental model (left). Simulation model in 3D (right)

The energy performance of a double-skin façade in general is influenced by the convective heat transfer coefficients, inside the cavity, which may vary depending on the strategy chosen for the ventilation (natural or mechanic) and depending also whether or not the solar radiation is present. The article is mainly focused on fluid flow and heat transfer, considering forced convection inside the cavity, during the summer, when venting superheated air is required, in order to maintain a good climate in the experimental room. The study highlights also the influence of the angle of the slats from the shading system on air flow, inside the cavity, by constructing 5 distinct cases: without the shading device and with shading device having the slats positioned at 0°, 25 °, 45 ° and 65 °. All the simulations are made in ANSYS 15 – Fluid Flow (CFX) Module and Steady State Heat Transfer Module.

2. General characteristics of experimental model

The experimental model to be presented in this study was built on the ground floor of the Civil Engineering Faculty in Brasov, on the south facade of the building. The model has an outer envelope consisting of a 10 mm secure glass in direct contact with exterior, placed at a distance of 1 m from an insulating double-glazed window 4-16-4 mm, which is the inner envelope. The outside air flows between the two glazing: during the winter the free convection transfers the solar heat gain to the interior space and during the summer through forced convection the superheated air is evacuated. Because of the greenhouse effect inside the cavity mechanical ventilation was required in order to extract the solar heat and evaluate this hot air flow that can be used for other purposes. The mechanical ventilation is provided by a duct fan VENTS TT150, common to the two input fresh air circuits, in the cavity, one in front of the shading system and the other behind it, as can be seen in Figures 1 and 2.

Box window double-skin façade and experimental room dimensions are presented in Table 1:

Table 1.

Box window double-skin façade and experimental room dimensions

Area	Length [mm]	Width [mm]	Height [mm]
Box DSF	1500	1000	2500
Experimental room	3600	2000	3000

To control daylight and solar radiation that enters the experimental room and improve working conditions for occupants, inside the cavity was installed in the middle area, a venetian horizontal blinds system, having shaped sides roll, made of special aluminum alloy enamel, 80 mm wide, UV resistant and weatherproof. The system is operated by means of a 220/240 V electric motor, 50 Hz, placed on the top of the shutter. The system has fully retractable blinds allowing maximum natural light in low light conditions and easy access for cleaning windows inside facade.

3. Fluid flow simulations

The fluid flow simulations presented in this paper concern only forced air circulation in the façade cavity, during the summer season. The purpose of fluid flow simulations is to determinate the medium cavity air velocity. As main input condition for simulations was used the measured air velocity in supply duct of the façade mechanical ventilation system, which was 4,5 m/s.

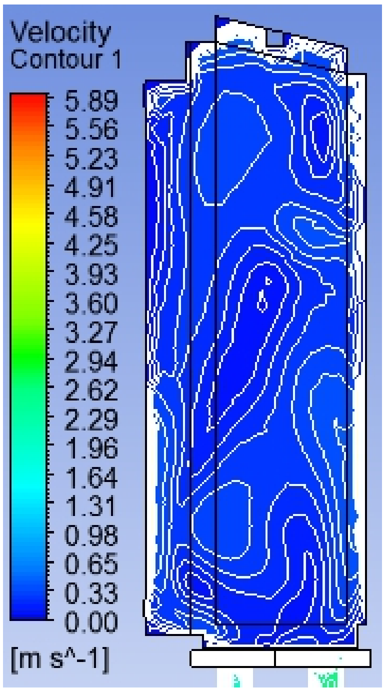
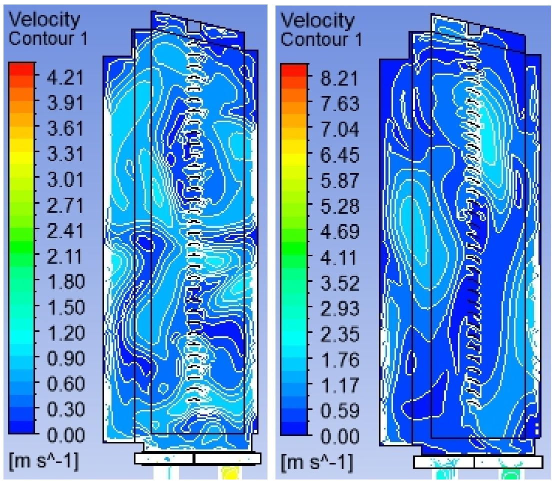
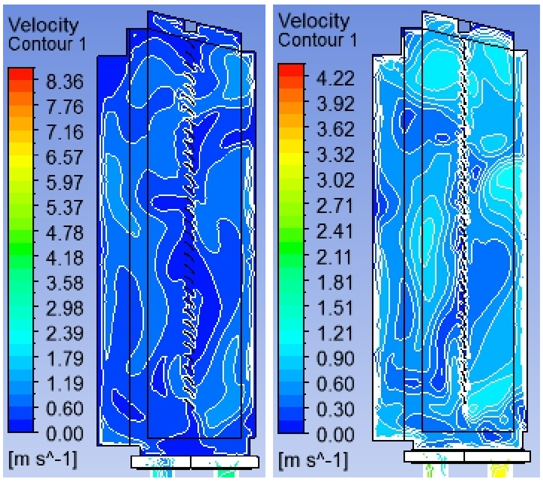


Fig. 2. Velocity contours inside cavity in case of shading system fully rose



a) b)

Fig. 3. Velocity contours inside cavity in case of shading slats positioned at 0° (a) and at 25° (b)



a) b)

Fig. 4. Velocity contours inside cavity in case of shading slats positioned at 45° (a) and at 65° (b)

Figures above show a mid-section plane with velocity contours, first in case of shading system fully raised (Figure 2) and then with shading slats positioned at 4 different angles 0°, 25°, 45° and 65° (Figures 3 and 4). If it's taken into consideration only the wide cavity area the medium velocities are those presented in table 2.

Table 2.

Medium velocities obtained

Case	I	0°	25°	45°	65°
Velocity [m/s]	0,26	0,52	0,71	0,54	0,62

As expected the smallest velocity is in the first case and all others are higher but all of them are less than 1 m/s.

More simulation work has still to be done in order to check the validity of models with respect of achieving more than 100 iterations and other operating conditions, i.e.

different outside/inside air temperatures as well as different values for solar radiation, this work being in progress.

4. Heat transfer simulations

The heat transfer simulations presented in this paper have been performed in steady state conditions, having as input conditions extreme climatic data for winter and summer. Boundary conditions for the two cases are presented below, in Figures 5 and 6.

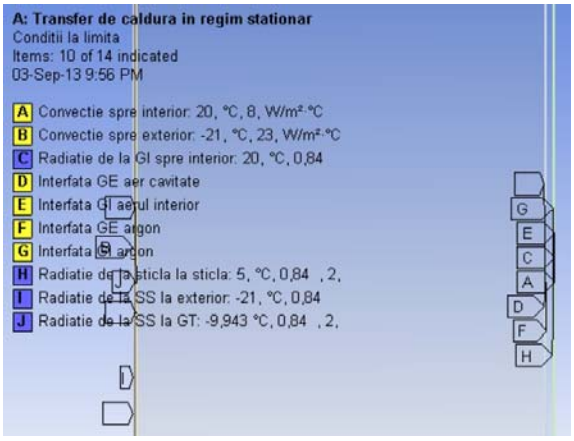


Fig. 5. Boundary conditions for the winter season

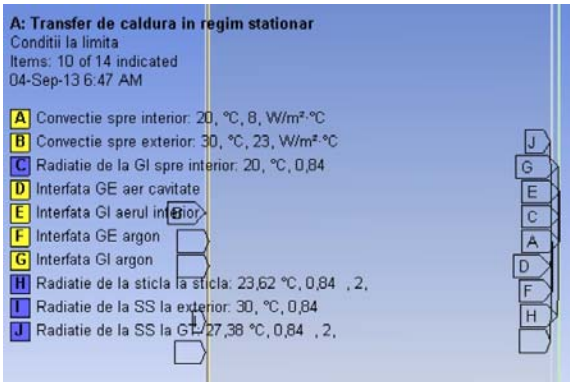


Fig. 6. Boundary conditions for the summer season

The results for heat transfer simulations are presented as temperatures and total heat fluxes for both cases winter and summer.

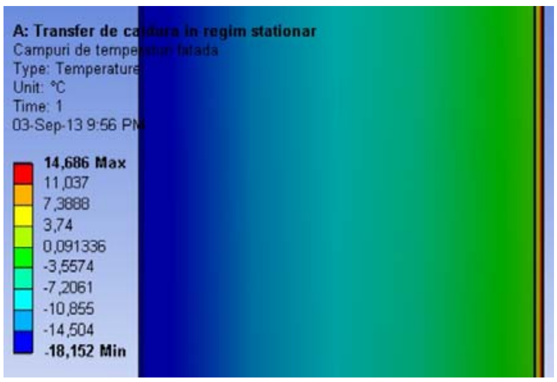


Fig. 7. Temperature distribution for winter season

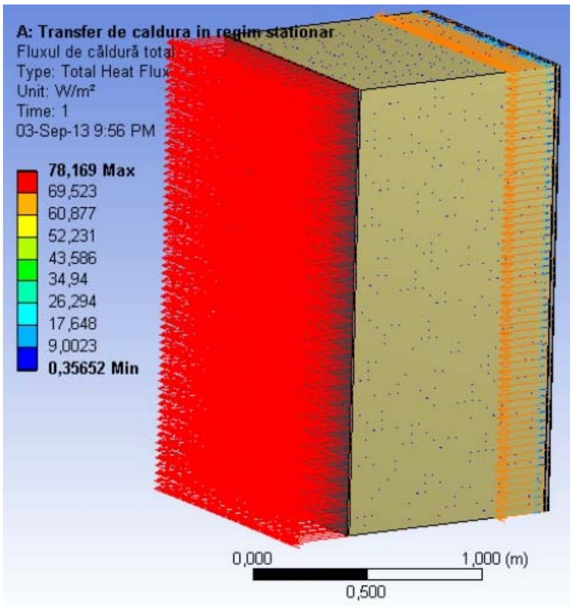


Fig. 8. Total heat flux for winter season

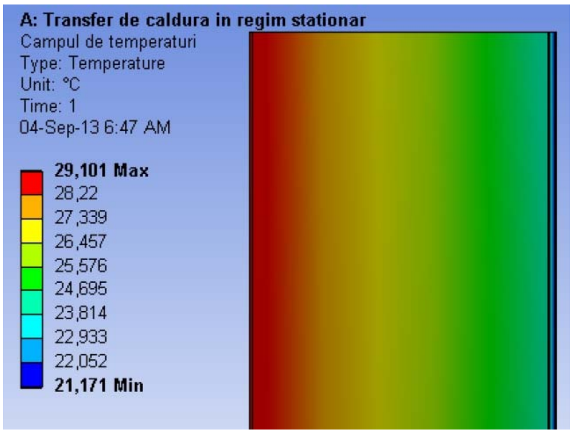


Fig. 9. Temperature distribution for summer season

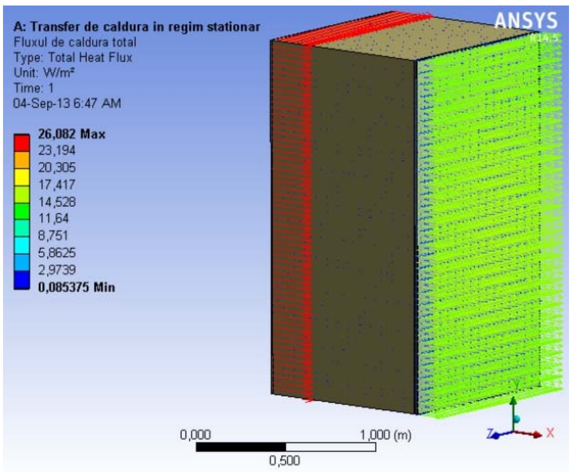


Fig. 10. Total heat flux for summer season

If it is to compare these results for heat transfer with those for a double pane window system (4 mm glass – 16 mm argon – 4 mm glass) keeping the same other conditions the results would be like those in Figures 11 and 12.

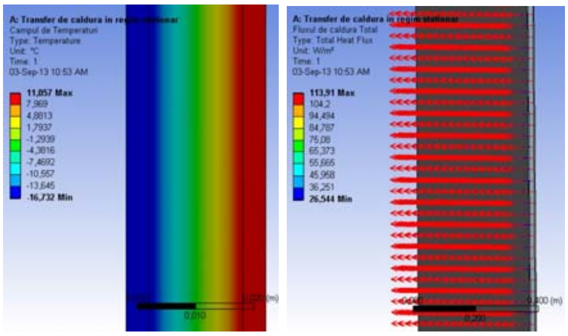


Fig. 11. Heat transfer simulation results for temperatures (left) and total heat flux (right), winter season in case of double pane window

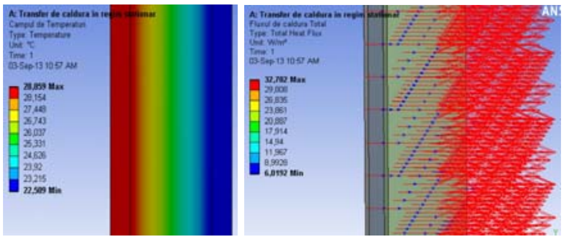


Fig. 12. Heat transfer simulation results for temperatures (left) and total heat flux (right), summer season in case of double pane window

If all figures above are analyzed we can say that in case of winter season the surface temperature inside experimental room is with 3,6°C higher in case of double-skin façade, comparing to the double pane window, this leading to a lower heat flux in case of double-skin façade. In this case the decrease in heat flux is about 31%.

In case of summer season the surface temperature inside experimental room is with 0,24°C higher in case if double pane window, comparing to the double-skin façade. In this case the decrease in heat flux is about 19%.

5. Conclusions

The complexity of the double skin façade considering elements which may vary from case to case depending on the location of the building, the local climate conditions, etc., the problem that arises in heat transfer is to evaluate the heat flux through the entire system, according to its geometric configuration, and the way the air flows through the cavity for different elements bounding the cavity i.e. their thermophysical properties.

These simulations are intended to be a starting point for modelling the same phenomenon with the entire double skin façade system, in transient condition and considering solar radiation. Such simulations should be run on a workstation in order to achieve over 600 iterations in a shorter period of time with more accurate results.

REFERENCES

Năstase G., Gavriliuc R., Şerban A.,– Box window double-skin façade. Experimental model in Brasov, Romania, *Advances in Environment Technologies, Agriculture, Food and Animal Science ISSN 2227-4359, 2013;*

Năstase G., Şerban A. – Box window double-skin façade. Heat transfer validation through inner envelope. Conferința CIBV 2013, ISSN 2065-2127, Editura Universității Transilvania din Brașov, 2013;

SR EN ISO 12631:2013 – Performanța termică a fațadelor cortină. Calculul coeficientului de transfer termic, *Asociația de Standardizare din România - ASRO, martie 2013;*

Năstase G., Gavriliuc R., Şerban A., Horneț M. – Office buildings with double-skin facade in Europe, Conferința CIBV 2012, ISSN 2285-7656, ISSN-L 2248-7648, Editura Universității Transilvania din Brașov, 2012;

Năstase G., Gavriliuc R. – "Transferul de căldură printr-un geam dublu tip termopan", Conferința Națională de Instalații, Sinaia 2012;

Adrian Bejan – "Convection Heat Transfer. Third edition", *Editura John Wiley & Sons, Inc., 2004*;

Florea Chiriac, Aureliu Leca, M. Pop, Adrian Badea, L. Luca, Nicolae Antonescu, D. Peretz - Procese de transfer de căldură și de masă în instalațiile industriale, *Editura Tehnică, 1982*;

E. Sandu, A.M. Bianchi, C. Mihăilă, V. Caluianu, N. Antonescu – Termotehnică și aparate termice, *Editura Didactică și Pedagogică, 1982*;

Ansys Design Modeler Training Manual, version 12, 1st Edition, *ANSYS INC, 2009*;

Ansys Design Modeler Workshop Supplement, version 12, 1st Edition, *ANSYS INC, 2009.*

ANSYS SIMULATION SOFTWARE, versiunea 14.5 – Steady State Thermal;

Ansys Design Modeler Training Manual, version 12, 1st Edition, ANSYS INC, 2009;

Ansys Design Modeler Workshop Supplement, version 12, 1st Edition, ANSYS INC, 2009;

Ansys Workbench – Mechanical Intro (Beta) Training Manual, version 12, 1st Edition, ANSYS INC, 2009;

Ansys Workbench – Mechanical Intro (Beta) Workshop Supplement, version 12, 1st Edition, ANSYS INC, 2009;

Ansys Meshing Application Introduction Training Manual, version 12, 1st Edition, ANSYS INC, 2009;

Kent L. Lawrence – ANSYS Workbench Tutorial, ANSYS Release 10, ISBN: 1-58503-269-7, SDC Publictions;

ANSYS Thermal Analysis Guide, Ansys release 10, ANSYS INC, august 2005;

ANSYS 14.5 HELP, ANSYS INC, 2013