HEFAT2010
7th International Conference on Heat Transfer, Fluid Mechanics and Thermodynamics
19-21 July 2010
Antalya, Turkey

NUMERICAL ANALYSIS OF AIR FLOW IN DOUBLE SKIN FAÇADE (DSF) WITH THERMAL MASS

Haifa el-sadi*, Imad Hmouz and Ali falahi
*Author for correspondence
TDE Techno Design
79-G BRUNSWICK Blvd.
DOLLARD DES ORMEAUX
QUEBEC, CANADA
H9B 2J5
E-mail: helsadi@tdetechno.com

ABSTRACT

This study was performed to analyze the impact of two parameters on the airflow: the velocity of air at the inlet and the type of the blind. The type of the blind has a major impact on the temperature distribution and on the velocity profiles in the cavity. On the other hand, thermal mass increase the energy performance of natural ventilation. The main objective is to generate the performance data of concrete thermal mass and their contribution to energy efficiency of the system by predicting thermal profiles in the double-skin façade, and the resulting cooling/heating loads for the adjacent perimeter interior zones during extreme summer/winter conditions.

INTRODUCTION

Many configurations of mechanically-ventilated DSF can exist in naturally-ventilated form as well (at least conceptually). An external air curtain double-skin façade was chosen as base-case configuration for study of naturally-ventilated DSF. External air curtain DSF: the supply air is from outside and the exhaust also returns back to outside. The driving force of airflow inside the cavity is due to buoyancy (stack effect) and wind force. The direction of airflow might be upward or downward according to outdoor condition. Exterior pane is reinforced single glazing, but interior pane is a double-glazed unit to avoid condensation by providing thermal insulation for warm indoor air against cold ventilation air in the gap during winter time. In heating season compared with single glazing it seems logical that addition of an extra layer results in an increase of the thermal resistance of the system. Furthermore, the incident solar energy can be better captured, which further decreases the transmission losses and even may provide gains. In cooling season the indirect solar gains captured in DSF components which is unwanted and can increase the cavity temperature is exhausted to outside. In this work, a base-case model for naturally-ventilated DSF(external air curtain) will be developed using building energy simulation software. it is capable of predicting the airflow rate inside the air channel and thermal distribution of DSF, simultaneously. The simulation results of base-case model will be verified.

Nomenclature

T	[K]	Temperature
Cp	[j/kgK]	Specific heat
i,j	[-]	two linked pressure nodes;
C_d	[-]	discharge coefficient commonly taken as 0.65
A	[m^2]	orifice opening area.
ρ	[kg/m3]	fluid density
P	[pa]	the static pressure
$\overset{=}{\tau}$	[pa]	the stress tensor
$\vec{g}$	[m/s^2]	the gravitational acceleration
x	[m]	Cartesian axis direction
Y	[m]	Cartesian axis direction

Zollner et al., 2002[1], conducted numerical and experimental studies in an external air circulation (both supply and exhaust from and to outside), naturally-ventilated DSF at the Technical University of Munich. J.Xaman et al. (2005) [2] studied numerically the fluid flow and heat transfer by natural convection in double skin façade using laminar and turbulent models. Li S. (2001) [3] reported that mechanically-ventilated DSF has 25% more cavity heat removal rate compared with naturally-ventilated DSF.

BOUNDARY CONDITIONS

To investigate on combined heat transfer and airflow of naturally-ventilated DSF, the measurement data from an outdoor test facility located at Technical University of Munich was used. It is an external air curtain DSF (air is supplied and exhausted from and to outside) toward south with dimension of 2.35m high, 0.9 width and 0.6m depth. The outer pane is a single glazing and inner a double glazing. The room attached to DSF has a depth of 3.1m. Aluminum venetian blind was installed in the gap between two panes 47cm from inner pane. As represented in Fig.1.

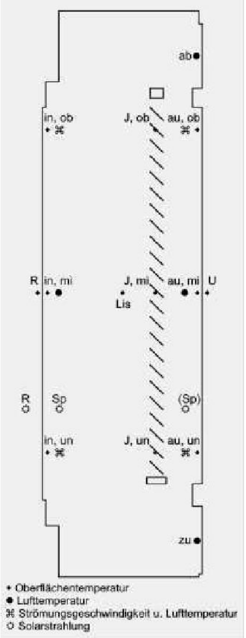


Figure 1 south-facing facade testing facilities for experimental investigation at Technical University of Munich. It includes stationary and variable testing façade.

The air flow within the double skin façade is considered steady state turbulent. For two-dimensional incompressible flows, the momentum and energy conservation equations are:

$$\frac{\partial u}{\partial x}+\frac{\partial \upsilon}{\partial y}=0 \qquad \textbf{(1)}$$

$$\nabla\,.\,(\rho\vec{\upsilon}\vec{\upsilon})=-\nabla p+\nabla.(\bar{\bar{\tau}})+\rho\vec{g} \qquad \textbf{(2)}$$

$$\nabla\,.\,(\rho\vec{u}T)=-\frac{1}{C_p}\nabla.(q) \qquad \textbf{(3)}$$

The turbulence is modeled using standard k-ε model. The radiation is modeled using the discrete ordinates radiation model (DO). Using Nodal Airflow Network, A common variation of the power law is related to the orifice equation:

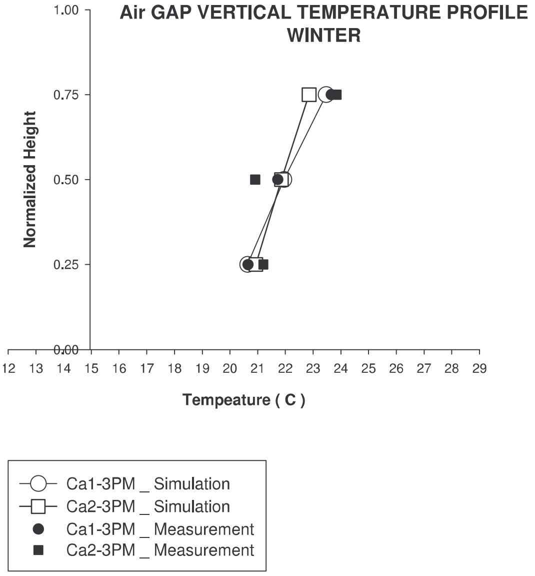
$$m_{i,j}=C_d\,A\sqrt{\frac{2\,(\Delta P_{i,j})}{\rho}} \qquad (4)$$

NUMERICAL METHOD

The CFD package FLUENT 6.3 is used to solve Navier-Stokes equations and turbulent flow. This CFD package uses the finite volume method. It enables the use of different discretization schemes and solution algorithms, together with various types of boundary conditions. As part of the same package, (a preprocessor) Gambit is used to draw the geometry and generate the required grid for the solver. An unstructured grid with triangle and quad elements is used. The analysis requires 1600 iterations for sufficient convergence. TRNSYS is a flexible simulation program primarily used in the fields of renewable energy engineering and multi-zone building energy simulation for passive as well as active solar design. Two airflow simulation approaches have been used in this simulation: Computational Fluid Dynamics (CFD) and the Network method. The Network method is of course much faster but only provide information about bulk flows. CFD on the other hand, provides details about the nature of flow field.

Validation of Numerical Models

Figure 2 show vertical temperature profiles of DSF. Generally temperature increases in vertical direction with increase of height. The reason is that the ventilated air which is coming from outside will have more time to contact with warmer surfaces. These warm surfaces include warm L3 surface at nighttime (next to room temperature which is warmer than outside) and hot venetian blind and glazings due to absorbed solar radiation.



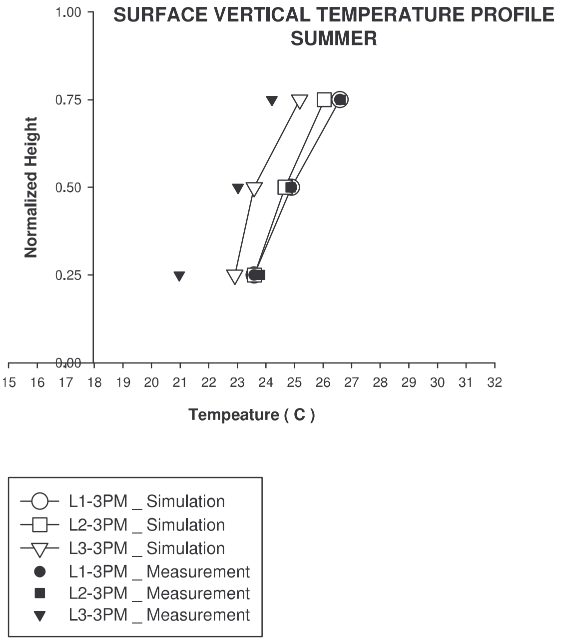


Figure 2 comparison of vertical temperature profile of simulated base-case and measurement data in a winter day; upper diagram (a) shows surface temperature and lower diagram (b) shows air gap temperature comparison. At winter-3PM.

PROCESSING OF RESULTS

It can be noticed from CFD analysis that the Al-blind and the thermal mass-concrete have the same profile. However, the temperature distribution is higher for the AL-blind than thermal mass-concrete. This is due to the material properties.

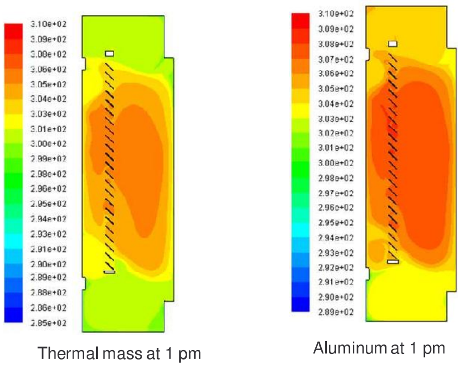


Figure 6. Temperature contours at 1 pm:
(a) thermal mass.
(b) Al-blind

Figure 3. Temperature contours at 1 pm:

(a) thermal mass.

(b) Al-blind

Figure 4 shows different contours of the velocities. It can be noticed that the flow turnings in the cavity around the blind increase with increasing the velocity. On the other hand, the increase in velocity is high; this is due to heat flux of the blind.

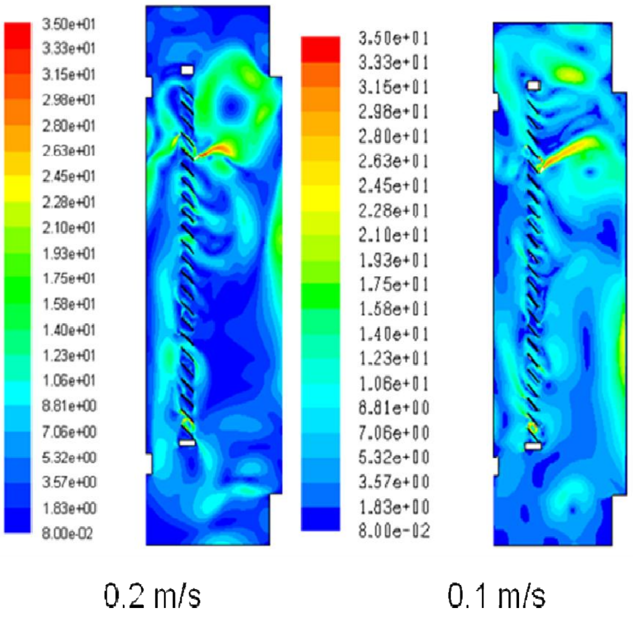


Figure 4: Contours of the Velocity of double skin façade-Al at 9 a.m, (a) 0.2 m/s, (b) 0.1 m/s

BUILDING ENERGY SIMULATION

TRNSYS is a flexible simulation program primarily used in the fields of renewable energy engineering and multi-zone building energy simulation for passive as well as active solar design. TRNSYS with its modular approach is able to interface with various multi-zone building airflow modeling programs such as CONTAM. Since Thermal mass can absorb, store heat and hold it for much longer periods of time than blind. Therefore, the energy saving is high by using DSF with thermal mass than using DSF with blind

CONCLUSION

Verification of base-case model results revealed that generally, there is a good agreement between measurements and simulation results in winter and summer. At night time in absence of solar radiation convection coefficient and long-wave heat transfer may cause errors which are less than day time. Between winter night-time and summer night-time, winter shows more deviation due to steeper temperature gradient with outdoor air. It can be concluded that the DSF with thermal mass in building has a unique energy saving advantage. The base-case model is able to predict thermal distribution and airflow rate of DSF under real operation conditions. The prediction of airflow model was restricted to bulk flow motion and detailed pattern of air movement was carried out by CFD.

REFERENCES

[1] Zollner, A., Winter, E.R.F., Viskanta, R., 2002. Experimental studies of combined heat transfer in turbulent mixed convection fluid flows in double-skin-façades. A. Int. J. Heat Mass Transfer 45 (22), 4401-4408.

[2] J. Xaman, G. Alvarez, L.Lira, C. Estrada, 2005. Numerical study of heat transfer by laminar and turbulent natural convection in tall cavities of facade elements. Energy and building journal 37, 787-794.

[3] Li S., A Protocol to Determine the Performance of South Facing Double Glass Façade System: A Preliminary Study of Active/Passive Double Glass Façade Systems, M.S. Thesis, Virginia Polytechnic Institute and State University, April 2001