Experimental and Computational Evaluation of the Thermal Performance of Double Skin Façades

M. Hernández T  
*University of Nottingham, UK*

L. Shao  
*De Montfort University, UK*

**ABSTRACT**

Double Skin Facades (DSF) have been a recently developed technology to improve the thermal performance of conventional façades of buildings which use large glazed areas. However, there has been a lack of test information on the behaviour and performance of a DSF. This is specifically the case when the façade has to perform under extreme or moderate summer conditions. In this case, the characteristics of thermal overheating and its practical control have not been subjected to systematic experimental and computational investigations.

This paper presents both experiments on a full-scale one-storey laboratory model of a Double Skin Façade and CFD simulations. The basic thermal behaviour in the façade cavity and the connected occupied space was investigated by a series of parametric studies and basic flow field investigations.

It was found that natural ventilation through a small number of moderately-sized façade openings is effective in significantly reducing the DSF overheating. A 3D model of the DSF chamber was made and modelled using CFD in order to visualise the airflow within the system. The objective was to study the airflow inside a DSF to complement the experimental work. The modelling work has demonstrated the feasibility and versatility of the technique for probing the flow and thermal behaviour of double skin façades.

**1. INTRODUCTION**

Large glazed areas are becoming a common issue in the design office buildings. The justification in the use of glass is mainly based on the argument that fully glazed envelopes contributes to improve natural lighting (Krewinkel, Heinz W., 1998). Nevertheless, these types of façades also add important heat gains and heat losses to the total energy employed for heating and cooling.

A Double Skin Façade is an envelope system, which have an external and internal layer that contains a buffer space used for controlled ventilation and solar protection (Oesterle, et al., 2001). The use of multilayered skins allows building insulation against thermal variations and external noise. The rate of heat transfer (U-Value) can be expressed in the case for a multiple glazing unit (Gan, G., 2001) by the following equation:

\[
U = \frac{1}{\frac{1}{h_c} + \frac{1}{h_i} + \frac{1}{h_t}}
\]

Where:
- \( U \) = W / m²K
- \( h_c \) = External heat transfer coefficient (W/m²K)
- \( h_i \) = Internal heat transfer coefficient (W/m²K)
- \( h_t \) = Conductance of multiple glazing units (W/m²K).

Double Skin Facades (DSF) reduce winter heating requirement. However, when the building is under summer conditions or located in moderate or hot climates; heat gains are predominant and the cost of cooling develops into a major issue (Faggenbauu, et al, 2002). The improvement on the system is necessary when working under hot climatic conditions. Previous research suggested that the use of ventilated facades makes progress on the energy reduction of indoor thermal gains (Costa, M. et al, 2000). The use of a ventilated channel reduces temperatures in the façade, though indoor thermal conditions have to be assessed in relation with the façade configuration as a part of the integration of the system with the building requirements.

The assessment of the thermal performance of a building is a complex task that requires detailed calculations of the different physical factors affecting the building. The assistance of CAD and thermal assessment tools are widely used for analysis of buildings on the design stages. The use of Computer Fluid Dynamic software (CFD) is increasingly employed as a powerful tool to predict the performance of buildings in specific conditions.

The aim of this study was to apply experimental testing and CFD to illustrate how the thermal behaviour under experimental conditions of a Double Skin Façade chamber contributes to give a better understanding of key elements that affects the thermal response and overheating of the façade under critical conditions.
2. EXPERIMENTAL WORK

2.1 Experimental Setup

The full-scale experimental chamber is illustrated on Figure 1. Its dimensions are 2,400 x 2,400 x 3,400 millimetres (W x H x D). The cavity dimensions are 2,400 x 2,400 x 800 mm (W x H x D). Considering an East-West orientation based on the radiation received in mid morning on vertical surface during mid season conditions at 30°N is 600 W/m² (CIBSE, 1986), the light rack was assembled with 12 flood halogen lights to simulate similar levels of radiation. Two main configurations of the DSF were used on the experimental tests: The Presence of Inlets/Outlet, which were called Standard Cases (ST) in which inlets and outlets are closed and The Standard Ventilated Cases (STV) in which the inlets and outlets are open. Within these main cases, the inlet/outlet size configuration, the DSF structure position and shading influence were also assessed.

2.2 Experimental Results

The air velocity inside the cavity was measured in one point for ST and STV. For the air flow behaviour, illustrated in Figure 2, there was a significant difference when inlets and outlets were closed in ST; the mean velocity reached was around 0.04 m/s. In contrast, higher mean velocity values of around 0.09 m/s were reached when inlets and outlets were open in STV.

Temperature differences (ΔT) within the cavity and the room in the ventilated cases are illustrated in Figure 3. The cavity reaches lower temperatures than the room when the outlets are open (STV), this is due to the constant air circulation favoured by stack effect. However, once the inlets are closed (ST), the cavity reaches most elevated temperature levels. As a result of this behaviour, the façade acts as an effective insulator against the external environment, proving the statement that it is an effective passive heating device and a significant key element for the cooling load due to the total energy transmittance to the interior (Manz, H., 2003). On the other hand, this performance illustrates the tendency to overheat when there is limited air movement.

The aperture size of inlet and outlets were also varied and the temperature values inside the cavity were measured (Fig 4). It was found that a reduction ratio of 50% on the outlet in relation with the inlet decreased the temperature inside the cavity by 3.3°C and also was found a similar thermal response when reducing the inlet size to 75% in relation with the outlet, the temperature inside the cavity was reduced by 2.8°C. This is probably due to...
aperture ratio of Inlet/Outlets contributing to increasing air speed/changes in air flow patterns, resulting in variation in reduction of temperatures inside the façade cavity. In summary, the following main issues were identified on the experimental tests:

- Lower cavity temperatures were obtained when the configuration is STV.
- Room temperatures are slightly lower in STV than in ST.
- Natural ventilation is effective reducing façade cavity overheating.
- The ratio of inlet and outlet apertures affects the temperature in the cavity.

The façade cavity acts as a thermal insulator against the external environment. Stratification of temperature levels within the cavity and the room was also found. There is a considerable reduction of room temperatures when shading is provided. Due to space limitation, results relating to these will not be presented here.

3. COMPUTER SIMULATION

The air flow in the DSF is calculated using the conservation equations for mass and momentum. As the flow also involves heat transfer, an additional equation for energy conservation is solved. Additional transport equations are also solved when the flow is turbulent. Computer Fluid Dynamics Software (FLUENT, 1993) is used to predict the flow and movement of air and temperature response of the laboratory model. Air turbulence is represented by the renormalization group (RNG) turbulence model developed by Yakhot and Orszag (Yakhot, et al., 1986), in which the incompressible steady-state flow time averaged equations are written as it follows:

$$\frac{\partial}{\partial x_i} \left( \rho U_i \phi \right) = \frac{\partial}{\partial x_i} \left( \Gamma \frac{\partial \phi}{\partial x_i} \right) + S_\phi$$

(2)

The flow model consists on the governing conservation equations of mass, momentum, heat transfer and turbulence, where:

- $\rho$ = Air density (kg/m$^3$)
- $U_i$ = Variable flow such as the mean velocity component $\xi$ (m/s) in $x_i$ (m) direction, pressure, temperature and turbulent parameters.
- $\Gamma \phi$ = Diffusion coefficient (N s/m²)
- $S_\phi$ = Source term.

Turbulence was calculated by standard k-ε model. In this case, Buoyancy-induced natural convection involves both laminar and turbulent flow. For this simulation, the model was built on separate cases for the complete model of the cavity in ST and STV.

3.1 Presence of inlet/outlet

The CFD models showed the predicted behaviour for the air movement and turbulence within the cavity, in which the airflow is fundamentally induced by buoyancy. Figure 5 shows how the airflow tends to behave from turbulent when the cavity is sealed to laminar when the cavity is ventilated, where high air velocities were reached.

Figure 5. Velocity magnitude contours of the cavity in ST and STV cases (m/s).

The total temperature values inside the cavity are shown in figure 6. The plot illustrates how there is a clear stratification of the temperature values.

Figure 6. Vertical temperature values in the centre of the cavity.

There is a very distinctive difference on the raise of temperature on each case; in the ST case the increment is reasonably steady and reaches high values, though the increment on the STV is variable due to displacement of the main flow towards the inner skin but reaching lower values than the sealed one, which makes evident the difference of temperature inside the cavity when is sealed or ventilated. For the airflow inside the cavity, Figure 7 also provides evidence of how there is continuous air movement due to the constant air flow from inlet to outlet. The higher velocity values were obtained at the same
height of inlet and outlets. When the cavity is sealed, the air velocity is more fluctuating with irregular but low magnitude values, which is due to vortices restrained in a confined space. It is clear that overheating develops when there is no constant airflow inside the DSF.

Figure 7. Air velocity values in the centre of the cavity.

3.2 Inlet/Outlet Size

The values of temperature obtained inside the cavity varied considerably according to the inlet/outlet size. The peak temperatures were reached using either inlet or outlet with size of 12.5mm. In these cases, the cavity on the simulation and the experimental models behave almost similar to the ST case. The lowest temperatures were obtained when there was an inlet/outlet size ratio of no more than 50%. These cases proved that a reduction on the opening size ratio of more than 50% has a direct influence on overheating development. The increment on the temperature values inside the cavity ranged by up to 16.5°C. It is greatest where inlet/outlet sizes were 12.5mm and smallest in the cases where the inlet/outlet ratio was not higher than 50%. The cavity temperature increased from the inlet to the outlet, where higher surfaces temperatures were reached due to the heat accumulated and hot air lifted by buoyancy. The air flow reaches higher velocities within the inlet and outlet where pressures are higher. The air velocity obtained inside the cavity was around 0.08 m/s, which is fairly close to the measured value using the experimental chamber.

DSF Structure Position

The air temperature values reached in the cavity when the amount of radiation was controlled on the external or internal skins were also calculated. The temperature levels obtained by the model are shown in Figure 10. The highest temperature levels were reached when the cavity was sealed and the internal skin was partially shaded. On the contrary, the lowest temperatures were obtained when the cavity was ventilated and 50% shaded on the external and the internal skin respectively. This also confirms that constant airflow is a key element on preventing overheating.

Figure 8. Air temperature values in the centre of cavity according to inlet/outlet size variation.

The values of temperature achieved by the model using shading devices – louvers - inside the cavity are illustrated in Figure 11. The values increased steadily when shading devices are used; however the temperatures reduced when reflective shading devices are used. Although absorptive of shading devices is directly related to the increase of temperature inside the cavity, the stack effect is also increased and can be used in favour of air removal from the facade or the building.

Figure 9. Air temperature in the centre of the cavity according to shading devices position.

Figure 10. Air temperature in the centre of the cavity according to shading devices position.

Figure 11. Cavity temperatures in the centre of the cavity according to shading devices colour.
3.4 Comparison of CFD and test results

Preliminary validation of the CFD modelling was performed by comparing steady state values temperature in three points of the facade test chamber with the ones obtained using the CFD model.

![Validation CFD Results: Cavity Openings](image1)

Figure 12. Temperature Values at three points for the experimental and simulated cases according the presence of inlet/outlets for the cavity.

Figures 12-13 show that relatively good agreement was obtained between the simulated and measured values, particularly for those on the cavity openings positions. The average difference between the predicted and measured temperatures was about 8.3%. The relative difference was larger on the simulation with shading devices. The result shows that CFD can be used to study the buoyancy-induced flow in open and closed cavities.

![Validation CFD Results: Shading devices](image2)

Figure 13. Temperature Values at three points for the experimental and simulated cases with shading devices.

4. CONCLUSIONS

The overheating of the cavity in the DFS includes basically the following issues:

- DSF encourages buoyancy, induces air movement, increases insulation and also allows radiation to flow inside the room.
- Convective forces inside the cavity could be used as an advantage to promote extraction of heat to minimise overheating.
- Natural ventilation is effective reducing overheating of the façade cavity.
- The size and ratio of inlet and outlet apertures affects the temperature and air flow in the cavity as do as shading devices.

This study confirms the suitability of CFD for DFS investigations, though its accuracy depends on setting correctly the boundary conditions. The mesh and geometry of the model affects the precision as well as the complexity of the CFD model. The conscious and responsible use of this tool is important to obtain useful data.

REFERENCES