Experimental and Numerical Investigation of a Mechanically Ventilated, Double Glazing Facade with Between-the-Panes Venetian Blinds

ABSTRACT

A BIPV/T system consisting of a mechanically ventilated, multi-skin façade, a between-the-panes venetian blind layer, and a between-the-panes photovoltaic (PV) array is considered. The combined photovoltaic/shading arrangement produces electricity and thermal energy in the form of preheated fresh air, and allows for adjustable daylighting. The aim of the study is to model the flow field and the temperature distribution around the venetian blind slats. Computational Fluid Dynamics (CFD) simulations were developed and validated at various blind slat angles. Temperature measurements were taken inside the ventilated cavity at various blind slat angles. Particle Image Velocimetry (PIV) measurements were made at zero slat angle only. The CFD simulations can predict the average blind layer and the indoor glazing layer temperatures within 2.3 and 5.4°C, respectively, of the experiment. The CFD simulations can predict the air peak streamwise velocities within 18% of the PIV measurements. The models can be used to obtain the distribution of heat flux and perform quick energy balance calculations. Between-the-panes heat transfer coefficients can be obtained to allow for future simple modelling and integration into building energy simulation software.

INTRODUCTION

Shading devices such as venetian blinds are used in building applications to control daylight, reduce glare, and to control the fenestration total thermal transmission (U-value) and Solar Heat Gain Coefficient (SHGC). They are commonly placed on the interior side of a glazing unit, but can also be placed in between a double- or multi-skin glazing arrangement.

There are several advantages of placing a blind layer in a between-the-panes arrangement. Other than portraying an aesthetically pleasing look, it facilitates the control and automation of the drive systems, because it provides a good location to safely place delicate mechanisms. It also brings additional solar-thermal savings if
ambient air is drawn from the outdoor environment, forced to flow over the blind layer, and then collected and supplied to the Heating, Ventilation, and Air Conditioning (HVAC) unit of building. This is what is referred to as a ventilated window in a supply-air mode.

Mechanically ventilated windows can be operated in a number of modes, depending on the positioning of the inlet and outlet with respect to the interior or exterior, and also the direction of air flow. Two of the most important modes are “supply-air” and “exhaust-air” because they can directly and effectively reduce the heating and cooling load of a building, respectively. A ventilated window, in fact, must be capable of operation under various modes to ensure overall year-round benefits. During the heating season, a supply-air arrangement forms an insulating air layer between the interior and exterior and also brings heated air inside the building. This compensates for the space heating load of the HVAC unit. During the cooling season, an exhaust-air arrangement can be used to alleviate overheating problems by taking the air inside the building, forcing it to flow upwards or downwards through the window and then exhausting it to the exterior. In doing so, surface temperatures inside the ventilated window will remain lower, and result in lower overall transmission gains during the summer.

The presence of a blind layer inside a ventilated window can significantly alter the thermal characteristics of a ventilated window. For example, the transmission gains and losses to the interior and the absorption gains, due to the convective motion of the air, all depend on the blind slat angle, spacings, material properties, and air flow rate.

Numerical models were developed and validated using data obtained from a roof-top experimental test hut at Concordia University (Figure 1). The experimental setup is equipped with a photovoltaic array, placed directly below the shading layer, and the combined PV-shading arrangement with the surrounding glazing and walls replaces an entire south-facing façade. Integrating a PV array with a ventilated window achieves a double gain in that it results in higher electricity generation efficiency and higher solar absorptin gains.

Numerical modelling of Building-Integrated Photovoltaic with Thermal (BIPV/T) systems is a challenging task due to the various modes of heat transfer and the coupled nature of the heat transfer mechanisms. Free and forced convection, long-wave and short-wave radiation, and conduction through solids must all be modeled. Further, conventional U- and g-value type models do not adequately represent the entire dynamics of the system because they do not take into account the enthalpy change of the air. In other words, the flow of heat is not only
in the horizontal direction, but is also in the vertical direction to the air. For these reasons, comprehensive CFD modelling, with as little simplifying assumptions as possible, must be considered as a first step in analyzing and optimizing BIPV/T's.

Once a validated CFD model is in place, it can be used to obtain within-cavity average heat transfer coefficients and pave way for simple nodal 1-D or 2-D modelling of the setup. Such simple models will be very helpful because they can be used to quickly and easily investigate the effect of a design change and can be easily integrated into a building energy simulation software. Preliminary results show that despite the complexity of the heat transfer interactions simple nodal based models, where every glazing, shading and air layer is represented by one node, are capable of modelling the thermal aspects of the system. Ongoing work is in progress to formulate empirical relations of within-cavity convective heat transfer coefficients that can be used in nodal-based models for design and optimization of BIPV/T systems.

Liao et al. (2007) and Charron and Athienitis (2006) studied this BIPV/T arrangement and concluded that the combined solar energy utilization efficiency of the system can reach as high as 60%. One-dimensional, two-dimensional, and CFD heat transfer models were developed for the lower section, consisting of PV, adjacent glazing and wall. To date, only the shading section of this arrangement is considered.

**EXPERIMENTAL SETUP**

The experimental setup (Liao et al. 2007) consists of a lower PV section and an upper shading section (Figure 1). In the lower section a Spheral-Solar™ PV panel was placed between a transparent glazing on one side and an opaque wall on the other. On the upper section, a venetian blind layer was placed mid-way between a double glazing unit on the interior side and a single-layer glazing on the exterior. The single-layer glazing is clear and 5 mm thick. The double glazing unit consists of 3-mm-thick glazing panes, spaced 12 mm apart, with a low-emissivity coating of value 0.1 applied on one side. Only the upper section of the Spheral-Solar™ configuration has been studied in this paper.

A variable-frequency fan, operating between a frequency range of 0–60Hz, forces air through the cavity. The cavity is 92–mm wide and the blind slats are 49–mm wide. Nine scenarios were tested corresponding to three blind slats angles (φ), 0, 45, and 75º, and three fan speed settings, 10, 20, and 30Hz on Sep 22, 2008 around noon (Table 1). During each scenario outdoor and indoor ambient temperature (T_{ext}, and T_{int}), wind, total
irradiation on a vertical surface \((I)\), and mean incoming (just below the lowest slat) air speed and temperatures 
\((\overline{V} \text{ and } T_{in})\), were measured, because they determine the boundary conditions for the CFD simulation. \(T_{in}, T_{ext}\), and \(I\) were measured using an integrated weather station. Incoming air velocity and temperature profiles were 
measured along a horizontal section just below the lowest slat, using a translating hot wire anemometer 
(temperature accuracy of ±0.3°C and velocity measurement repeatability of 0.03 m/s ±1% of reading). \(\overline{V}\) and \(T_{in}\) 
were calculated from anemometer readings through:

\[
\overline{V} = \frac{w}{\int_{0}^{w} \rho dx} \int_{0}^{w} \rho dx \\
\overline{T}_{in} = \frac{w}{\int_{0}^{w} \rho V dx} \int_{0}^{w} \rho V dx
\]

Table 1

<table>
<thead>
<tr>
<th>Scenario</th>
<th>Fan Speed</th>
<th>(\phi)</th>
<th>(\overline{V}) m/s</th>
<th>(T_{in}) °C</th>
<th>(T_{ext}) °C</th>
<th>(T_{int}) °C</th>
<th>(I) W/m²</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>10</td>
<td>0</td>
<td>0.13</td>
<td>40.5</td>
<td>13.0</td>
<td>22.8</td>
<td>680</td>
</tr>
<tr>
<td>2</td>
<td>20</td>
<td>0</td>
<td>0.31</td>
<td>36.8</td>
<td>13.0</td>
<td>22.8</td>
<td>680</td>
</tr>
<tr>
<td>3</td>
<td>30</td>
<td>0</td>
<td>0.56</td>
<td>32.7</td>
<td>13.0</td>
<td>22.8</td>
<td>680</td>
</tr>
<tr>
<td>4</td>
<td>10</td>
<td>45</td>
<td>0.12</td>
<td>34.4</td>
<td>10.8</td>
<td>22.0</td>
<td>650</td>
</tr>
<tr>
<td>5</td>
<td>20</td>
<td>45</td>
<td>0.35</td>
<td>32.7</td>
<td>10.8</td>
<td>22.0</td>
<td>650</td>
</tr>
<tr>
<td>6</td>
<td>30</td>
<td>45</td>
<td>0.62</td>
<td>29.1</td>
<td>10.8</td>
<td>22.0</td>
<td>650</td>
</tr>
<tr>
<td>7</td>
<td>10</td>
<td>75</td>
<td>0.13</td>
<td>39.3</td>
<td>13.1</td>
<td>22.2</td>
<td>692</td>
</tr>
<tr>
<td>8</td>
<td>20</td>
<td>75</td>
<td>0.46</td>
<td>35.3</td>
<td>13.1</td>
<td>22.2</td>
<td>692</td>
</tr>
<tr>
<td>9</td>
<td>30</td>
<td>75</td>
<td>0.64</td>
<td>30.8</td>
<td>13.1</td>
<td>22.2</td>
<td>692</td>
</tr>
</tbody>
</table>

During each scenario, radiation-shielded T-type thermocouples (accuracy ±0.5°C) were used to measure the 
blind temperature \((T_b)\) and the outdoor and indoor glazing temperatures \((T_{g,o} \text{ and } T_{g,i})\) at three locations inside 
the ventilated channel: close to the top frame \((11^{th} \text{ slat from the bottom})\), center-window \((7^{th} \text{ slat from the bottom})\), and close to the bottom frame \((3^{rd} \text{ slat from the bottom})\). \(T_{b}\) was measured, for each slat, on various top 
and bottom and shaded and unshaded spots on that slat.

PIV measurements were taken at zero blind slat angle only (scenarios 1, 2, and 3). The blind slat angle was 
set to zero for all PIV measurements to limit undesirable laser beam blockage behind the slats. At non-zero slat 
angles, the shaded regions behind the slats are large and result in inaccurate velocity calculations. To measure
non-zero slat angles, a more sophisticated laser-mounting apparatus, such as a special tripod, must be used to
direct the laser beam parallel to the slats.

PIV is a non-intrusive method of obtaining the velocity vector map of a fluid. It is based on the spatial and
temporal resolution of motion of small particles released inside the flow. These particles, called seeding
particles, act like markers in that comparing the location of them at the beginning and at the end of a short time
interval, the velocity vector of the fluid can be calculated at that specific time and location. A complete PIV
system, including the laser and the computer processor, was installed inside the test hut. The laser is of type
Nd:YAG with a laser-sheet thickness of 1.5 mm. A 10-bit camera (1600x1186 pixels) and 60mmf/2.8D lens was
used to capture the PIV pictures. Olive oil particles (approximately 3 micro-meter diameter) were used as
seeding material. The time between the pulses ranged from 650 µs to 2600 µs, depending on the maximum flow
speed, to ensure appropriate particle displacement within an interrogation cell. For each measurement, about 500
sets of pictures were taken. The plane of measurement consisted of a section containing three consecutive blind
slats and the space in between. Figure 2 presents the positioning of the PIV equipment. While the laser and the
camera are normally placed perpendicular to each other, they were placed parallel to each other, using a 45˚
reflecting mirror, because limited space was available on the edge of the window. An adaptive cross-correlation
algorithm (FLUENT, 2005) was employed to calculate velocity vectors in a final interrogation cell size of 32 ×
32 pixels with 50% overlapping between adjacent cells.

**LAYER TEMPERATURE STRATIFICATIONS**

Considering the temperature data, the temperature stratification along every glazing/shading layer can be
very large at times. The blind layer slat surface temperatures can vary as much as 12.8°C along the height and
between shaded and unshaded spots (the blind slat solar reflectivity is 0.59). The blind slats may sometimes
cause shading on parts of their neighboring slats. The blind slat temperature can vary by as much as 2–2.5°C
when measured on a single slat depeing on whether the thermocouples are shaded or not. The inner-glazing
layer had a maximum temperature variation of about 5°C, but the outer-glazing layer temperatures were roughly
uniform along the height. The cavity air temperature can vary by as much as 5–6°C from inlet to outlet, during
low flow rate and high irradiation scenarios. An air temperature stratification of such magnitudes suggests that
significant savings can be realized in winter heating load of a building.
NUMERICAL SIMULATION SETUP

Numerical simulation of BIPV/T’s is typically carried out using an solar-thermal separation technique, meaning that the solar-optical properties of each layer and solar absorption by each layer are calculated separately in a sub-simulation, which feeds into a main CFD thermal simulation. The sub-simulation provides with heat generation terms that are used in the main CFD simulation. The CFD simulation accounts for long-wave and convective exchange, but not solar radiation exchange.

The solar absorption sub-simulation developed in this work was based on a layer-type analysis of glazing arrays. A shading array can be included in the analysis of glazing arrays provided that its effective, or spatially-averaged, properties are used. The model allows for the computation of the fraction of incident radiation absorbed, reflected, or transmitted through every shading/glazing layer (Wright and Kotey 2006). The effective solar-optical properties of each layer, accounting for multiple reflections/transmissions, were used to track the beam and diffuse fluxes though the glazing/shading array. The effective properties of the glazing layers depend on material, thickness, and angle of incidence. The effective properties of the venetian blind layer depend on the slat surface reflectivity, slat angle, profile angle, and the geometry (Kotey and Wright, 2006). Once the effective properties of each layer and also the incoming beam and diffuse fluxes are known, a set of linear algebraic equations can be solved to give the absorbed fluxes in each layer.

All flows encountered in this study are subjected to large strain rates and significant swirls due to presence of very large flow dividers (i.e. the blind slats). The Reynolds’ numbers based on channel width range from 1400 to 6000. Although the critical Reynolds’ number in literature for channel flow is 2300, the flow was modelled as turbulent at all times because the dividers were suspected to cause an early onset of turbulence. The realizable $k$–$\varepsilon$ model, developed by Shih et al. (1995), has been selected to treat turbulence. The realizable $k$–$\varepsilon$ model has a revised dissipation rate equation and a revised eddy viscosity formulation, over the standard $k$–$\varepsilon$ model. The eddy viscosity depends on the mean strain rates, rotating rates, angular velocity of the system rotation, and the turbulence fields. The realizable $k$–$\varepsilon$ model has been extensively validated for channel flows, rotating and separating flows, and flows over obstacles (Shih et al. 1995; S.E. Kim et al. 1999; FLUENT 2005). The realizable $k$–$\varepsilon$ has been previously employed for similar channel-type flows in BIPV/T systems (Liao et al. 2007; Safer 2005) at Reynolds’ numbers as low as 1600.
The flow was modelled as steady and two dimensional. The computational domain consisted of the double-glazing unit, the ventilated cavity, the blind slats and the single-glazing unit. Refer to Figure 3 for a schematic of the computational domain. Blind slats were modelled as flat (curvature effect neglected) and having zero-thickness. Long-wave radiation was solved between the surfaces, including inside the double-glazing unit, using a grey-diffuse radiation model treating air as a non-participating medium. Buoyancy-driven effects were also solved in both sealed and ventilated cavity. The Boussinesq approximation was used for the variation of density (FLUENT, 2005).

A two-layer based model was adopted treating the viscosity affected region and the core turbulence region individually. In the near the wall region the viscous sublayer was resolved using an Enhanced Wall Treatment option, and in the core turbulent region the realizable k- and ε- equations were applied. The realizable k–ε model has a revised dissipation rate equation and a revised eddy viscosity formulation over the standard k–ε model. Unlike the standard k–ε model the eddy viscosity model parameter, $C_p$, is not constant (Shih et al., 1995, and FLUENT, 2005). The conservation equations are applied iteratively (Gauess-Seidel) until convergence is achieved. The conservation equations for mass, X-momentum, Y-momentum, energy, $k$, and $\varepsilon$ equations and the eddy viscosity formulation are as follows:

\[
\frac{\partial}{\partial x} (\rho u) + \frac{\partial}{\partial y} (\rho v) = 0 \quad (3)
\]

\[
\frac{\partial}{\partial x} (\rho u u) + \frac{\partial}{\partial y} (\rho v u) = -\frac{\partial P}{\partial x} + \frac{\partial}{\partial x} ((\mu + \mu_t) \frac{\partial u}{\partial x}) + \frac{\partial}{\partial y} ((\mu + \mu_t) \frac{\partial v}{\partial y}) \quad (4)
\]

\[
\frac{\partial}{\partial x} (\rho v v) + \frac{\partial}{\partial y} (\rho u v) = -\frac{\partial P}{\partial y} + \rho g \beta (T - T_{ref}) + \frac{\partial}{\partial x} ((\mu + \mu_t) \frac{\partial v}{\partial x}) + \frac{\partial}{\partial y} ((\mu + \mu_t) \frac{\partial u}{\partial y}) \quad (5)
\]

\[
\frac{\partial}{\partial x} (\rho u T) + \frac{\partial}{\partial y} (\rho v T) = \frac{\partial}{\partial x} \left( \frac{K + K_t}{c_p} \frac{\partial T}{\partial x} \right) + \frac{\partial}{\partial y} \left( \frac{K + K_t}{c_p} \frac{\partial T}{\partial y} \right) \quad (6)
\]

\[
\frac{\partial}{\partial t} (\rho k) + \frac{\partial}{\partial x_i} (\rho k u_j) = \frac{\partial}{\partial x_i} \left[ \left( \mu + \mu_t \right) \frac{\partial k}{\partial x_i} \right] + G_k + G_b - \rho \varepsilon \quad (7)
\]

\[
\frac{\partial}{\partial t} (\rho \varepsilon) + \frac{\partial}{\partial x_j} (\rho \varepsilon u_j) = \frac{\partial}{\partial x_j} \left[ \left( \mu + \mu_t \right) \frac{\partial \varepsilon}{\partial x_j} \right] + \rho C_1 S \varepsilon - \rho C_2 \frac{\varepsilon^2}{k + \sqrt{\varepsilon}} + C_{1e} \frac{\varepsilon}{k} C_{2e} G_b \quad (8)
\]
\[ \mu_t = \rho C_\mu \frac{k^2}{\varepsilon} \]  \hspace{1cm} (9)

where

\[ C_i = \max \left[ 0.43, \frac{\eta - 5}{\eta + 5} \right], \eta = S \frac{k}{\varepsilon} \]  \hspace{1cm} (10)

\[ C_{1\varepsilon} = 1.44, C_2 = 1.9, \sigma_k = 1.0, \sigma_\varepsilon = 1.2 \]  \hspace{1cm} (11)

The pressure-velocity coupling was solved using the SIMPLE algorithm (FLUENT, 2005). Momentum, energy, \( k \) and \( \varepsilon \) equations were solved using a First Order Upwind scheme. The under-relaxation factors for pressure, momentum, \( k \) and \( \varepsilon \) were set to 0.3, 0.7, 0.8, and 0.8. A non-uniform mix of triangular and quadrilateral elements was used with increasing mesh density near the fluid-solid boundaries to allow for the Enhanced Wall Treatment of the boundaries. About 35000 control volumes were used to discretize the computational domain (1.96 mm control volume size). The convergence criteria were less than \( 10^{-4} \) for all equations and less than \( 10^{-7} \) for the energy equation.

**Boundary Conditions**

The air velocity and temperature profiles, hydraulic diameter, and turbulent intensity are specified at the inlet. The incoming air velocity and temperature profiles are based on the anemometer measurements. The hydraulic diameter is taken to be twice the width of the channel. The turbulent intensity is set equal to 10% always. The outlet is specified as having zero gauge pressure. Solid regions, i.e., the blind and the glazings, have specified volumetric heat generation terms. Air with constant properties at 20°C, except density, is assigned to fluid regions, i.e., the closed cavity and the channel.

The outdoor convective heat transfer coefficient ranged from 5 to 7.4 W/m²K and was calculated based on the outdoor wind speed, \( W \), and the cubic root of building volume, \( Z \) (Duffie and Beckman, 1991):

\[ h_{\text{out}} = \max[5, 8.6W^{0.6}/Z^{0.4}] \]  \hspace{1cm} (12)

The indoor convective heat transfer coefficient was calculated from the correlation suggested in ASHRAE 2005, where \( \Delta T \) stands for the difference in surface temperature and indoor air, and \( H \) stands for the height of the window (ASHRAE, 2005). The indoor convective heat transfer coefficient varied from about 2 to 3 W/m²K.

\[ h_{\text{in}} = 1.46(\Delta T/H)^{0.25} \]  \hspace{1cm} (13)
VELOCITY VALIDATION

Instantaneous and time-averaged velocity vector fields from PIV reveal that the general shape of the velocity fields agrees with the CFD simulations. Instantaneous velocity vector fields reveal that, at all blind slat angles and fan speeds, the flow tends to move mostly on the left and the right sides of the blind slats (i.e., the core flow region) and two relatively slow-moving vortices are observed between the slats, which change in size and location with time. An example of an instantaneous velocity vector field is presented in Figure 4. The fluid velocity is relatively high in the core flow region and no apparent mass crosses over the slats (from one side of the blind layer to the other). The contours of streamlines obtained from CFD revealed that the flow speed is relatively high in the core flow region and two counter-rotating vortices of about equal size are formed filling up the entire space between every two blind slats (Figure 5). These vortices have been previously observed at $\varphi = 0^\circ$ in a numerical study of flow field around blind slats (Safer et al., 2005). Although the mean flow speeds and the ratio of the slat width to the channel width were quite different in that work, similar vortices were observed. The blind layer effectively divides the flow into two streams, one to the left and one to the right of the blind layer, where no apparent mass cross-over occurs between the two streams.

While instantaneous velocity fields suggest that the vortices change in size and location, time-averaging the velocity fields tends to give seemingly large and stable-looking vortices, of about equal size, filling up the entire space between every two blind slats (Figures 6–8). Therefore, the average velocity fields seem to compare better with the CFD solutions, qualitatively, than the instantaneous fields. Although the average velocity fields seem to match with the CFD solutions, neither of the two gives a true representation of what actually happens in the flow. From the instantaneous fields, unlike what the average fields suggest, the flow is likely unsteady and periodic because the vortices are not stable.

In the regions near the glazings (boundary layer regions) and near the blind slats (especially near the tips of the slats) undesirable laser scattering causes much contrast difference in the PIV pictures compared to the surrounding regions. The undesirable laser scattering causes the outlines of the glazing and the blind slats to be visible in Figures 6–8. In the regions near the glazings and blind slats, the PIV calculations are not reliable and must not be used for any sort of validation; However, in the core flow region and in between the blind slats (regions not very close to the glazings or the blind slats), both average and instantaneous fields show fast uni-
directional motion and some sort of slow recirculatory motion, respectively, suggesting that the general shape of
the velocity field agrees between CFD and PIV.

Considering the streamwise velocities, the streamwise velocity profiles match with the CFD predictions in
the recirculation regions and in the core flow regions, except near the glazings. The streamwise velocity profiles
are presented in Figures 9 and 10, corresponding to scenarios 1 and 3, at a horizontal section, \( y = 475.6 \) mm,
mid-height between slats 8 and 9. In the region where largest errors occur (to the right of the blind layer in the
core flow region the error) is about 33% at 10 Hz (absolute error of 0.12 m/s) and 21% at 30 Hz (absolute error
of 0.32 m/s). While these errors seem to be large, peak velocities (in the core flow regions on the right and left
side of the blind layer) compared well between PIV and CFD. At 10 Hz the peak velocities have an error of 5%
on the left side and 18% on the right side of the blind layer, and at 30 Hz they have errors of 11% on the left and
0.5% on the right side.

**TEMPERATURE VALIDATION**

To validate the CFD model thermodynamically, area-averaged glazing and shading layer temperatures
(averaged over the entire height of the layers) were compared against average layer temperautre measurements
(averaged over the three measuring locations along each layer) (Table 2). The experimental temperature results
presented in Table 2 have a confidence interval of 95%. The uncertainties in these results were calculated
according to

\[
\bar{T} - \sigma_T < T < \bar{T} + \sigma_T
\]  

and

\[
\sigma_T = \sqrt{b^2 + p^2}
\]

where

\( \bar{T} \)  Mean temperature

\( \sigma_T \)  Uncertainty in mean temperature (95% confidence interval)

\( b \)  Bias error: ±0.5°C according to the thermocouple manufacturer

and
\[ p = \frac{1.96\sigma}{\sqrt{N}} \]  

(16)

where

\( p \) Precision error

\( \sigma \) Standard deviation

\( N \) Number of samples (N > 10)

<table>
<thead>
<tr>
<th>Scenario</th>
<th>Fan Speed Hz</th>
<th>( \phi ) deg</th>
<th>( T_{b, \text{CFD}} ) °C</th>
<th>( T_{b, \text{Exp}} ) °C</th>
<th>( T_{g,i, \text{CFD}} ) °C</th>
<th>( T_{g,i, \text{Exp}} ) °C</th>
<th>( T'_{g,i, \text{Exp}} ) °C</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>10</td>
<td>0</td>
<td>57.0</td>
<td>59.9</td>
<td>44.1</td>
<td>41.7</td>
<td>0.74</td>
</tr>
<tr>
<td>2</td>
<td>20</td>
<td>0</td>
<td>47.4</td>
<td>47.7</td>
<td>38.6</td>
<td>38.5</td>
<td>0.87</td>
</tr>
<tr>
<td>3</td>
<td>30</td>
<td>0</td>
<td>40.1</td>
<td>40.5</td>
<td>34.2</td>
<td>36.4</td>
<td>1.0</td>
</tr>
<tr>
<td>4</td>
<td>10</td>
<td>45</td>
<td>50.5</td>
<td>49.2</td>
<td>39.5</td>
<td>38.4</td>
<td>0.87</td>
</tr>
<tr>
<td>5</td>
<td>20</td>
<td>45</td>
<td>40.7</td>
<td>40.4</td>
<td>34.5</td>
<td>35.2</td>
<td>0.79</td>
</tr>
<tr>
<td>6</td>
<td>30</td>
<td>45</td>
<td>35.6</td>
<td>34.9</td>
<td>30.5</td>
<td>32.5</td>
<td>0.86</td>
</tr>
<tr>
<td>7</td>
<td>10</td>
<td>75</td>
<td>56.1</td>
<td>57.2</td>
<td>46.0</td>
<td>45.4</td>
<td>0.62</td>
</tr>
<tr>
<td>8</td>
<td>20</td>
<td>75</td>
<td>44.9</td>
<td>47.2</td>
<td>36.5</td>
<td>41.9</td>
<td>1.0</td>
</tr>
<tr>
<td>9</td>
<td>30</td>
<td>75</td>
<td>40.8</td>
<td>39.6</td>
<td>35.0</td>
<td>37.6</td>
<td>1.1</td>
</tr>
</tbody>
</table>

Indoor (and outdoor) glazing temperatures are strongly affected by indoor (and outdoor) heat transfer modelling and are not as well predicted. But the blind temperatures, which are strongly affected by the irradiation levels and convective heat transfer modelling around the slats, agree well between measurement and simulation. The blind surface temperatures can vary by as much as about 2°C across the width of a single slat between the shaded and unshaded spots.

Considering the distribution of heat fluxes, the heat fluxes that reach the interior space are of utmost importance for a building energy designer. These heat fluxes which contribute to the gain of the system are the transmitted portion of solar radiation, the convective and radiative fluxes from the inner-most glazing layer to the interior space, and the ventilated air enthalpy change per unit area of window. These heat fluxes have been calculated from the simulation and presented in Figure 11.

Because the ventilated cavity brings preheated air to the window (hotter than the exterior and usually hotter than even the interior) and also because the blind slats have a large area available for solar absorption (blind slat width is 49 mm), the direction of heat transfer is usually from the inner-most glazing to the interior, not the other
way around. In other words, the mechanically ventilated BIPV/T can reduce the transmission losses, and even produce transmission gains. The convective and long-wave radiative fluxes to the interior are combined into one, and the sum of the two is presented in Figure 11. The ventilated air enthalpy change is due to the absorbed fluxes of solar radiation, and to a lesser extent, due to the recovery of a portion of transmission losses or gains. The heat removal rate is very high, in magnitude, compared to the convective and radiative transmission gains and transmitted solar radiation entering the building.

Results indicate that by increasing the air flow rate, surface temperatures tend more toward the outdoor ambient temperature, and as a result, the magnitude of the convective and radiative heat fluxes from the window to the interior space fall. The air heat gain however increases substantially from 10Hz, to 20Hz, and to 30Hz at all $\phi$. At $\phi = 75^\circ$ since the blind slats are in an almost-closed position the portion of the solar radiation passing through the entire setup and reaching the interior space, the transmitted solar flux is very small. Decreasing $\phi$, at any given fan setting, seems to induce less flow rate because the effective width of the channel through which the flow is moving is decreases.

Increasing the air flow rate can significantly enhance the convective heat transfer inside the cavity, resulting in increased air thermal gains. Therefore a higher fraction of absorbed solar radiation is removed by the air, instead of flowing to interior or the exterior environment. The air thermal gain is indeed so significantly affected by the flow rate that under some circumstances it is practical to increase the flow rate to improve the building heating demand, despite higher transmission losses or lower transmission gains and higher fan pressure drop losses. There is an upper limit on the flow rate however based on the stability of the blind layer and the noise problems.

**CONCLUSIONS**

An overview of the experimental and numerical results for a BIPV/T system were presented. The velocity fields at practical flow rates showed great similarity at $\phi = 0^\circ$. The flow field was periodic, from one slat to another, and relatively very slow-moving counter-rotating vortices were observed between every two blind slat. CFD model were developed and validated against the experimental data at three different blind slat angles, $\phi = 0$, 45 and 75°. The glazing and shading surface temperatures were predicted fairly well by CFD. The glazing temperatures were very sensitive to indoor and outdoor heat transfer modelling, but the blind temperatures were
less sensitive to them. Large temperature stratifications were observed inside the cavity. The air temperature difference between inlet and outlet can reach as high as 5-6°C at very high irradiation levels and very low-speed fan settings. The presence of mechanical ventilation enhances the convective heat transfer inside the cavity. Air thermal gains increased, rather significantly, by increasing the flow rate while the transmission gains through the window decreased. Air thermal gains were much larger than the transmission gains and they increased so significantly with rising flow rates that it makes it practical at times to increase the flow rate to achieve better overall gains.

ACKNOWLEDGEMENTS

This work was partially funded by the Canadian Solar Buildings Research Network, a strategic NSERC (Natural Sciences and Engineering Foundation of Canada) research network and Natural Resources Canada.

REFERENCES

**Figure 1**  The experimental set up at Concordia, plane of PIV measurements and placement of thermocouples

**Figure 2**  Placement of PIV equipment inside the test hut.
Figure 3  The computational domain

- $w = 92 \text{ mm}$
- $H = 530 \text{ mm}$
- $t_{g1} = 3 \text{ mm}$
- $t_{g2} = 3 \text{ mm}$
- $t_{a0} = 5 \text{ mm}$
- $w_s = 49 \text{ mm}$
- $t_s = 0.17 \text{ mm}$
- $s = 43.9 \text{ mm}$
- $\phi = 0, 45, \text{ and } 75^\circ$

Figure 4  The instantaneous velocity field, $\phi = 0^\circ$, fan speed = $30 \text{Hz}$, $\nabla = 0.56 \text{ m/s}$.
Figure 5  The streamline instantaneous velocity field, $\phi = 0^\circ$, fan speed = 30Hz, $\nabla = 0.56$ m/s.

Figure 6  Approximate streamline contours and velocity vector field, $\phi = 0^\circ$. Fan speed = 10Hz, $\nabla = 0.13$ m/s.
Figure 7  Approximate streamline contours and velocity vector field, $\phi = 0^\circ$. Fan speed =20Hz, $\vec{V} =0.31$ m/s.

Figure 8  Approximate streamline contours and velocity vector field, $\phi = 0^\circ$. Fan speed =30Hz, $\vec{V} =0.56$ m/s.
Figure 9  Streamwise velocity profile from PIV and CFD, mid-height between slats 8 and 9, \( \phi = 0^\circ \), 10 Hz, \( \overline{V} = 0.13 \text{ m/s} \).

Figure 10  Streamwise velocity profile from PIV and CFD, mid-height between slats 8 and 9, \( \phi = 0^\circ \), 30 Hz, \( \overline{V} = 0.56 \text{ m/s} \).
Transmitted Solar Flux
Convective and Radiative Heat Flux
Air Thermal Gain

<table>
<thead>
<tr>
<th>$\Phi = 0$</th>
<th>$\Phi = 45^\circ$</th>
<th>$\Phi = 75^\circ$</th>
</tr>
</thead>
</table>

**Figure 11**  Solar and thermal Gains