ABSTRACT
The present work is an experimental and numerical study of a Building-Integrated Photovoltaic/Thermal (BIPV/T) system located at Concordia University. The experimental setup consists of a multi-skin glazing array with a Venetian blind and photovoltaic device located in the glazing cavity. Ambient air is drawn through the setup. As air moves upwards it is heated by the blind and glazing layers, and the PV panel. The entire system produces electricity and thermal energy in the form of preheated fresh air, and allows for adjustable daylighting.

This paper presents the modeling of air flow and temperature distribution around the blind slats in order to help with the design and optimization of such BIPV/T systems. Particle Image Velocimetry (PIV) and temperature measurements were taken. A Fully-coupled 2-D Computational Fluid Dynamics (CFD) model was developed to study the flow field and temperature field. The velocity field was predicted accurately by the model. The surface temperatures obtained by simulation, although sensitive to indoor and outdoor heat transfer modeling, agreed well with the experimental recordings.

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 the blind layer in such 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 a form of mechanical ventilation is utilized.

In the present work a BIPV/T system consisting of a ventilated transparent cavity with an internal venetian blind has been studied. The experimental setup is located at the rooftop test facility at Concordia University in Montreal (Fig. 1). A photovoltaic panel has been placed directly below the blind layer, producing electricity, and the combined PV-shading arrangement with the surrounding glazing and walls replaces an entire south-facing façade. Integrating a shading section with a lower PV section under mechanical ventilation achieves a double gain in that it results in higher electrical efficiency and also provides the building with a source of preheated fresh air.

Liao, Charron and Athienitis (Liao et al., 2007 and Charron, 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- and two- dimensional 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 data was obtained for three different fan speeds. PIV measurements of the velocity field and temperature measurements were taken at various locations within the setup. Ambient conditions were also recorded.

A numerical model was developed and validated against the experimental results. The numerical model consists of a CFD model, in commercial code FLUENT, and a solar-absorption sub-model to calculate absorbed solar fluxes in the glazing/shading array.
EXPERIMENTAL SETUP
The experimental setup (Liao et al., 2007) consists of a lower PV section and an upper shading section (Fig. 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 half-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.

A variable-frequency fan, operating between a frequency range of 0 – 60 Hz, forces air through the cavity. The cavity is 92 mm wide and the blind slats are 49 mm wide. T-type thermocouples were placed along the blind layer, and glazing surfaces directly adjacent to ventilated cavity fluid (Fig 1). Thermocouples were also placed just below the lowest slat to give the inlet air temperature profile used by the numerical model. The ambient air temperature, wind speed, and wind direction were recorded every minute. Total incident solar radiation was recorded by a pyronometer placed on the façade. Beam incident radiation was measured by a Normal-Incidence Pyrheliometer.

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

A PIV system was installed inside the test hut for the purpose of velocity measurements. A Nd:YAG-type laser was used with a laser sheet thickness set to 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 measurements was chosen at a location about midway along the shading section containing three blind slats and the space in between. Figure 1 presents the plane of measurements, and Figure 2 presents the positioning of the PIV equipment. An adaptive cross-correlation algorithm was employed to calculate velocity vectors in a final interrogation cell size of 32 × 32 pixels with 50% overlapping between adjacent cells.

EXPERIMENTAL RESULTS

Five scenarios corresponding to three different fan speeds, 8, 15Hz, and 30 Hz, were tested in Mar 28, 2007, resulting in mean flow speeds of roughly 0.1 to 0.6 m/s. Due to the fluctuations in outdoor wind speed and due to the changing thermal conditions over the course of the day of the experiment (such as solar radiation), the mean flow speeds were slightly different for scenarios when the fan speed was the same. The blind slat angle was set to zero for all cases. The incidence and profile angles were calculated for each scenario from standard formulae. The sky was clear during all scenarios.

Post Processing of the PIV data revealed a great deal of similarity among all scenarios. The shape and the qualitative characteristics of the velocity field were similar, despite varying flow rates and thermal boundary conditions. The majority of the flow was moving only on the left or right side of the blind slats. Between the blind slats, two recirculation zones of relatively slow-moving fluid, and of about equal size, were observed. Refer to Figure 3 for the velocity field obtained from PIV data corresponding to scenario 3. These recirculation zones have been previously observed in a numerical study of flow field around blind slats by Safer et al., 2005. For the cases studied in that work, the cavity width to slat width ratio and mean flow speeds were quite different but similar counter-rotating vortices were observed.

Considering the temperature data, the variation of temperature along the blind and inner-glazing layer was rather large. A maximum of 12.8 °C rise of surface temperature was observed along the blind layer (the blind slat solar reflectivity is 0.59), and 6.8 °C along the inner glazing. The outer-glazing temperatures, however, were roughly uniform along the height. Table 1 summarizes the mean temperature readings as well as the vertical variation in temperature.

<table>
<thead>
<tr>
<th>Scenario</th>
<th>T&lt;sub&gt;Tgo&lt;/sub&gt; (°C)</th>
<th>ΔT&lt;sub&gt;Tgo&lt;/sub&gt; (°C)</th>
<th>T&lt;sub&gt;Tb&lt;/sub&gt; (°C)</th>
<th>ΔT&lt;sub&gt;Tb&lt;/sub&gt; (°C)</th>
<th>T&lt;sub&gt;Tgi&lt;/sub&gt; (°C)</th>
<th>ΔT&lt;sub&gt;Tgi&lt;/sub&gt; (°C)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Scenario 1</td>
<td>34.0</td>
<td>1.7</td>
<td>41</td>
<td>3.6</td>
<td>26.0</td>
<td>4.7</td>
</tr>
<tr>
<td>Scenario 2</td>
<td>27.5</td>
<td>3.1</td>
<td>23.7</td>
<td>4.0</td>
<td>22</td>
<td>1.5</td>
</tr>
<tr>
<td>Scenario 3</td>
<td>44.5</td>
<td>1.2</td>
<td>61.2</td>
<td>12.8</td>
<td>38.5</td>
<td>6.8</td>
</tr>
<tr>
<td>Scenario 3</td>
<td>35.6</td>
<td>1.7</td>
<td>42.4</td>
<td>5.9</td>
<td>31.5</td>
<td>3.7</td>
</tr>
<tr>
<td>Scenario 4</td>
<td>26.2</td>
<td>2.2</td>
<td>23.3</td>
<td>5.2</td>
<td>23.5</td>
<td>1.1</td>
</tr>
</tbody>
</table>

Table 1: Temperature readings and their respective vertical variation along the shading/glazing layers.
Figure 3: The velocity vector map pertaining to scenario 3.

SOLAR ABSORPTION BY EACH ELEMENT

A layer-based analysis was undertaken, allowing for the computation of the fraction of incident radiation absorbed, reflected, or transmitted through every shading/glazing layer (Wright, 2006). The effective solar-optical properties of each layer, accounting for multiple reflections/transmissions, were used to track the beam and diffuse fluxes through 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, 2006). Table 2 summarizes the absorbed fluxes at each layer.

### Table 2: Summary of Solar Absorption results

<table>
<thead>
<tr>
<th>Scenario</th>
<th>Time</th>
<th>Slat Angle (deg.)</th>
<th>Profile Angle (deg.)</th>
<th>( I_{\text{net}} ) ( (\text{W/m}^2) )</th>
<th>Abs. ( (\text{W/m}^2) ) 5-mm glazing</th>
<th>Abs. ( (\text{W/m}^2) ) 3-mm clear glazing</th>
<th>Abs. ( (\text{W/m}^2) ) 3-mm low-e glazing</th>
</tr>
</thead>
<tbody>
<tr>
<td>Scenario 1</td>
<td>15 Hz 10:45</td>
<td>0</td>
<td>51.2</td>
<td>615</td>
<td>101</td>
<td>288</td>
<td>20.5</td>
</tr>
<tr>
<td>Scenario 2</td>
<td>30 Hz 12:00</td>
<td>0</td>
<td>46.5</td>
<td>718</td>
<td>118</td>
<td>318</td>
<td>20.5</td>
</tr>
<tr>
<td>Scenario 3</td>
<td>8 Hz 13:15</td>
<td>0</td>
<td>47.4</td>
<td>754</td>
<td>123</td>
<td>327</td>
<td>30.9</td>
</tr>
<tr>
<td>Scenario 4</td>
<td>15 Hz 14:00</td>
<td>0</td>
<td>48.7</td>
<td>732</td>
<td>120</td>
<td>323</td>
<td>28.6</td>
</tr>
<tr>
<td>Scenario 5</td>
<td>30 Hz 14:30</td>
<td>0</td>
<td>45.4</td>
<td>700</td>
<td>113</td>
<td>313</td>
<td>27.7</td>
</tr>
</tbody>
</table>

**NUMERICAL MODELING**

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 Re numbers based on channel width range from 1400 to 6000. The flow was modelled as turbulent for all cases because the dividers were suspected to cause an early onset of turbulence. The realizable k-\( \varepsilon \) model has been selected because it is suitable for rotating flows undergoing separation and recirculation (FLUENT Manual). The realizable k-\( \varepsilon \) model has been previously employed by Liao, 2007, Charron, 2006, Safer, 2005, and Posner, 2003, for similar channel-type flows with large dividers and at Re 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, blind slats and the single-glazing unit. Refer to Figure 4 for a schematic of the computational domain and boundary conditions. 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.

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 \( \varepsilon \)- equations were applied. The conservation equations for the Realizable k-\( \varepsilon \) model are as follows:

Continuity conservation equation:
\[
\frac{\partial}{\partial x} (\rho u) + \frac{\partial}{\partial y} (\rho v) = 0
\]

(1)

\[\text{x-momentum conservation equation:} \]
\[
\frac{\partial}{\partial x} (\rho u^2) + \frac{\partial}{\partial y} (\rho uv) = -\frac{\partial P}{\partial x} + \frac{\partial}{\partial x} \left( \mu \frac{\partial u}{\partial x} \right) + \frac{\partial}{\partial y} \left( \mu \frac{\partial u}{\partial y} \right)
\]

(2)

\[\text{y-momentum conservation equation:} \]
\[
\frac{\partial}{\partial x} (\rho v^2) + \frac{\partial}{\partial y} (\rho uv) = -\frac{\partial P}{\partial y} + \rho g \beta (T - T_{ref}) + \frac{\partial}{\partial x} \left( \mu \frac{\partial v}{\partial x} \right) + \frac{\partial}{\partial y} \left( \mu \frac{\partial v}{\partial y} \right)
\]

(3)

Energy conservation equation:
\[
\frac{\partial}{\partial x} (\rho c_T) + \frac{\partial}{\partial y} (\rho c_T v) = \frac{\partial}{\partial x} \left( K \frac{\partial c_T}{\partial x} \right) + \frac{\partial}{\partial y} \left( K \frac{\partial c_T}{\partial y} \right)
\]

(4)

\[\text{Turbulent kinetic energy transport equation:} \]
\[
\frac{\partial}{\partial t} \left( \frac{k}{\rho} \right) + \frac{\partial}{\partial x} \left( \frac{j}{\rho} \right) = \frac{\partial}{\partial x} \left[ \mu + \frac{k}{\sigma_k} \frac{\partial k}{\partial x} \right] + G_k + G_b - \rho \varepsilon - Y_M + S_k
\]

(5)

\[\text{Turbulent dissipation rate equation:} \]
\[
\frac{\partial}{\partial t} \left( \frac{\varepsilon}{\rho} \right) + \frac{\partial}{\partial x} \left( \frac{j}{\rho} \right) = \frac{\partial}{\partial x} \left[ \mu + \frac{k}{\sigma_k} \frac{\partial \varepsilon}{\partial x} \right] + \rho C_1 S_k \varepsilon + C_2 \left( \frac{\varepsilon^2}{k + \sqrt{K} h} \right) + C_3 \frac{\varepsilon}{k} G_k + S_e
\]

(6)

where
\[
C_1 = \max \left[ 0.43, \frac{\eta}{\eta + 5} \right], \eta = \frac{S k}{\varepsilon}
\]

\[
C_{ik} = 1.44, C_2 = 1.9, \sigma_k = 1.0, \sigma_\varepsilon = 1.2
\]

(7)

(8)

The pressure-velocity coupling was solved using SIMPLE (FLUENT Manual). 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. The convergence criteria were less than \(10^{-4}\) for all equations and less than \(10^{-7}\) for the energy equation.

**Boundary Conditions**

At inlet the velocity and temperature profiles, as well as turbulent intensity and hydraulic diameter were specified. The no-slip boundary condition was applied at all walls. The inner-most and outer-most walls were subjected to convection and long-wave radiation exchange with surrounding. The sealed cavity walls of the double-glazing unit were insulated at ends. The outlet was specified as having zero gauge-pressure.

The outdoor convective heat transfer coefficient ranged from 5 to 7.4 W/m\(^2\)K and was calculated based on the outdoor wind speed, \(W\), and the cubic root of building volume, \(Z\) (Duffie, 1991):
\[
h_{conv, out} = \max[5.8, 6.6 W^{0.6} / Z^{0.4}]
\]

(9)

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):
\[
h_{conv, in} = 1.46 (\Delta T / H)^{0.25}
\]

(10)

![Figure 4: The computational domain, and boundary conditions.](image)

**NUMERICAL RESULTS**

The velocity field obtained by the CFD simulation agrees well with the experimental results in that recirculation zones of same type and size are observed, at all times, between the blind slats. Also, the flow seems to have a periodic pattern from one slat to another for all flow speeds (Figure 5).

Figure 6 presents the streamline velocity at a section half-way between two adjacent blind slats for both
CFD and PIV results. Scenario 3 is chosen as an example. A comparison between the CFD and PIV graphs reveals that they intersect the x-axis at about the same distance, indicating that the size of the recirculation zones are predicted well by the CFD model. Also, the CFD model predicts the streamwise velocity well at every location within the cavity, with a maximum relative error of 32%, except near the right and left glazing boundaries where unwanted laser beam scattering occurs from the glazing surfaces.

Mid-cavity temperature values ($T_{go,2}$, $T_{gl,2}$ and $T_b,2$ in reference to Fig. 1) are summarized in Table 4, and compared against their respective experimental recordings. The results generally agree with the experimental results. The blind slat temperature seems to be predicted well. The glazing temperatures are very sensitive to the indoor and outdoor boundary conditions and are not as well predicted. For better prediction of glazing temperatures more accurate correlations/models for obtaining the indoor and outdoor heat transfer coefficient are recommended. The air temperature difference from inlet to outlet reaches a maximum of roughly 4 to 5 °C under practically minimum flow rate and maximum incident solar radiation (near solar noon, Scenario 3). An increase in air temperature of such magnitudes can bring significant savings in winter heating load of building.

![Figure 5: The streamline contours corresponding to Scenarios 3, 4 and 5.](image)

![Figure 6: Comparison of streamwise velocity profile between PIV and CFD results at a mid-section between two adjacent blind slats ($y = 376.15$ mm).](image)
<table>
<thead>
<tr>
<th>Scenario</th>
<th>$V_{in}$ (m/s)</th>
<th>$T_{in}$ ($^\circ$ C)</th>
<th>$I_{tot}$ (W/m$^2$)</th>
<th>$T_{go}$ Exp ($^\circ$ C)</th>
<th>$T_{go}$ CFD ($^\circ$ C)</th>
<th>$T_{b}$ Exp ($^\circ$ C)</th>
<th>$T_{b}$ CFD ($^\circ$ C)</th>
<th>$T_{gi}$ Exp ($^\circ$ C)</th>
<th>$T_{gi}$ CFD ($^\circ$ C)</th>
<th>$T_{out}$ CFD ($^\circ$ C)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Scenario 1</td>
<td>0.19</td>
<td>29.0</td>
<td>615</td>
<td>34.0</td>
<td>26.0</td>
<td>41</td>
<td>40.5</td>
<td>29.7</td>
<td>31.5</td>
<td>32.8</td>
</tr>
<tr>
<td>Scenario 2</td>
<td>0.580</td>
<td>21.0</td>
<td>718</td>
<td>27.5</td>
<td>22</td>
<td>23.7</td>
<td>28.5</td>
<td>17.9</td>
<td>21</td>
<td>23.2</td>
</tr>
<tr>
<td>Scenario 3</td>
<td>0.11</td>
<td>46.2</td>
<td>753</td>
<td>44.5</td>
<td>38.5</td>
<td>61.2</td>
<td>62</td>
<td>45.5</td>
<td>49</td>
<td>51.2</td>
</tr>
<tr>
<td>Scenario 4</td>
<td>0.17</td>
<td>32.9</td>
<td>732</td>
<td>35.6</td>
<td>31.5</td>
<td>42.4</td>
<td>47</td>
<td>33.1</td>
<td>36</td>
<td>40.2</td>
</tr>
<tr>
<td>Scenario 5</td>
<td>0.495</td>
<td>20.6</td>
<td>700</td>
<td>26.2</td>
<td>23.5</td>
<td>23.3</td>
<td>28</td>
<td>19.5</td>
<td>22</td>
<td>22.9</td>
</tr>
</tbody>
</table>

CONCLUSIONS

An overview of the experimental and numerical results for a BIPV/T system were presented. The velocity field showed great resemblance for practical mean flow speeds. The flow field was periodic, from one slat to another, and relatively very slow-moving counter-rotating vortices were observed among all scenarios. The CFD model was validated against the experimental data. The temperature profile along the shading layer was predicted well by the model, but the glazing temperatures were not as well predicted, owing to the high sensitivity of glazing temperatures to indoor and outdoor convective heat transfer coefficients. To predict the temperature along the glazing layers more accurately, more sophisticated models for indoor and outdoor heat transfer are needed. The air temperature along the upper section of cavity rose by about 4 to 5 $^\circ$C at maximum incident solar radiation suggesting that significant savings in winter heating load may be realized using such glazing/shading arrangements under forced ventilation.

ACKNOWLEDGMENTS

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

NOMENCLATURE

Abs  The absorbed flux in each layer  
Exp  Experimental results  
h$_{conv,in}$  Convective heat transfer coefficient between the inner glazing and indoor ambient air  
h$_{conv,out}$  Convective heat transfer coefficient between the outdoor glazing and outdoor ambient air  
I$_{tot}$  Total incident solar radiation on a vertical surface  
$T_{in}$  Mean inlet temperature  
$T_{out}$  Mean outlet temperature  
$T_{go}$  Outer-glazing surface temperature directly adjacent to the ventilated cavity fluid  
$T_{gi}$  Inner-glazing surface temperature directly adjacent to the ventilated cavity fluid  
$T_{b}$  Blind slat surface temperature  
$T_{ambient,in}$  Indoor ambient air temperature  
$T_{ambient,out}$  Outdoor ambient air temperature  
$V_{in}$  Mean inlet velocity

REFERENCES


FLUENT 6.3 Manual.

Safer N., Woloszyn M., Roux J.J., 2005, “Three-dimensional simulation with a CFD tool of the airflow phenomena in single floor double-skin...

