NUMERICAL MODEL DEVELOPMENT OF A PV-INTEGRATED DOUBLE FAÇADE

Chris Hadlock¹, Mike Collins¹ and Andreas Athienitis²

¹Department of Mechanical Engineering, University of Waterloo, 200 University Avenue West, Waterloo, ON Canada, N2L 3G1
Phone: 519-888-4567 ext 3885, Fax: 519-885-5862, Email: cjhadloc@uwaterloo.ca
²Department of Building, Civil & Environmental Engineering, Concordia University, 1515 Sainte-Catherine St. West, Montreal, QC Canada, H3G 2W1
Phone: 514-848-2424 ext 8791, Fax: 514-848-7965

ABSTRACT

A two-dimensional numerical model has been developed to model an airflow window (AFW) with building-integrated PV (BIPV). The entire system, often referred to as a BIPV with thermal generation, or BIPV/T, can generate both electricity and heat for a building. The BIPV/T system of interest uses on forced convection, and not buoyancy-induced flow. The present work examines only the AFW section, consisting of a double-glazed façade with between-the-panes roller blind. The lower section of the BIPV/T system, consisting of the BIPV portion, has already been modelled at Concordia University. This paper presents the steps required for the development of the numerical model and discusses some of the challenges associate with this process. The numerical results are compared to experimental data obtained from the outdoor test facility at Concordia University.

INTRODUCTION

Today, more than ever before, architects, engineers, builders and the academic community are working together to help make windows more energy efficient. As a result, we have witnessed the emergence of new technologies such as: substitute fill gases like argon and krypton, spectrally selective coatings, tinted glazings, switchable glazings, multi-paned windows, etc. Of particular interest is the emergence of the airflow window (AFW) and building-integrated photovoltaics (BIPV), which have recently been amalgamated to form a unique system which can help meet electrical, thermal and lighting requirements of a building.

The concept for the AFW, which emerged in the UK around the 1860’s, has been gaining credibility of late, especially in Central Europe (Manz et al., 2004) where many new commercial buildings are making use of the technology. The main functions of the AFW are to generate thermal energy and to provide some form of daylighting control. Much like AFW’s, PV panels are also being integrated into building facades. BIPV is now adorning entire building facades, replacing typical building materials with components capable of generating electricity to be used by the building. The coupling of the AFW with the BIPV, called a double façade with BIPV or a building-integrated photovoltaic/thermal (BIPV/T) system, is a relatively new idea; very little literature can be found on BIPV/T systems. Literature on the individual components that make up the system are more prevalent. Modeling of mechanically ventilated double facades has been investigated by many (e.g. Ciampi et al., 2003 and Hensen et al., 2002) as has ventilated BIPV (Mei et al., 2003).

Several advantages can be realized with a BIPV/T system, especially for the Canadian climate where 60% of the total residential energy and 52% of the total commercial energy consumed goes to space heating (NRCan, 2003). With the additional heat created by the PV panel, the thermal efficiency of the system is increased. This removal of heat from the PV also increases the electrical efficiency of the solar panel as its efficiency is adversely affected by a rise in temperature. As a result of combining these two systems together, a simultaneous increase in the efficiency of both the thermal and electrical system is achieved. Between-the-panes shading devices, such as slats, can be used to stimulate additional heat absorption to the air. Research by Ye et al. (1999) has shown the effects of slat angle on heat transfer coefficients. In a different study, Rheault et al. (1989) found that up to 36% of the HVAC load required for a Canadian winter climate could be saved using automated blind systems.

A collaboration between Concordia University and the University of Waterloo is presently underway which aims to optimize BIPV/T systems. However, before system optimization can be attempted, it is imperative that the thermal and fluid dynamics be characterized. Various feasible options exist for such an endeavor. In the early stages of this project, an analytical model for the BIPV section of the BIPV/T system was developed (Charron, 2004). This model was intended to provide insight and direction for subsequent models. After the completion of the analytical model, a numerical model was developed for the BIPV section (Liao, 2005). The work presented is a continuation of the aforementioned
models. The objective of this work is to characterize the AFW portion of the BIPV/T system as seen in Figure 1. Once both the AFW and BIPV models have been developed, the next step will be to perform a building energy simulation. Software such as TRNSYS can be used to perform a transient building energy analysis (e.g. Mei et al., 2003). In this study, Fluent was used to develop a numerical model for the characterization of the heat transfer and fluid flow within the upper section of the BIPV/T, comprising two glazings with an internal roller blind.

EXPERIMENTAL SETUP

At Concordia University, the Solar Lab consists of two different BIPV/T systems which have been installed on a common building located outside on the roof of a four story building in downtown Montreal, Quebec. The two different BIPV/T systems are distinguished by their respective PV cells. One of the systems is equipped with a Photowatt solar panel while the other employs a Spheral Solar panel to generate its electricity as seen in Figure 2. The Photowatt side uses a roller blind as a shading device while the Spheral side uses a venetian blind. Both shading devices are located within the airflow cavity between the double glazings. Both systems rely on forced convection, created by fans whose frequency is controlled digitally from inside the Solar Lab. Although the air speed through the cavity is controlled by a fan, the effects of wind are definitely present. And although the effects of wind on double facades has been the topic of numerous experimental investigations (e.g. Saelens et al., 2001), a good predictive model is not easily attainable. In an attempt to limit the affects of wind on the airflow, dampers have been used at the inlets of both BIPV/T configurations. Both the indoor and outdoor conditions, including the solar gain, air temperature, relative humidity, pressure, wind speed, etc. were calculated for all experiments using equipment described by Liao (2005). Temperature data, which was obtained from T-type thermocouples and recorded in one-minute intervals by the data acquisition system, was accompanied by particle image velocimetry (PIV) equipment, used to record the corresponding velocity data. A full description of these systems have been described Liao (2005).

NUMERICAL MODELING

The numerical modeling of the entire BIPV/T system was accomplished by subdividing the system into two parts, the upper section, consisting of the AFW, and the lower section where the BIPV was located (Figure 2). The upper section, comprising two glazings, one between-the-pane roller blind, an inlet and an outlet, was modeled at the University of Waterloo using Fluent. The inner and outer glazings are both 21 mm thick. A low-E coating is located on the outer surface of the interior glazing (i.e. on the surface in contact with the air cavity). The between-the-panes roller blind measures 0.75 mm in thickness and is made of a double-weave vinyl pattern. The dimensions of the AFW section are 1 m x 1 m x 0.092 m (L x H x W). Due to the geometrical configuration, the airflow through the cavity can be modelled as two-dimensional providing end effects are ignored. By modeling the system as two-dimensional, the glazings and blind layer could be modeled as parallel plates with the appropriate thicknesses as well as optical and thermal properties. A two-dimensional view of the cavity can be seen in Figure 3.

MATHEMATICAL MODELS

Fluent, which uses a control volume analysis, applies the basic conservation equations to each cell within the flow domain. For a two dimensional incompressible flow, the following steady-state mass conservation equation is used

\[ \frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} = 0 \]
where \( u \) and \( v \) are the velocity components in the x- and y-direction, respectively as depicted in Figure 3.

\[ \phi_x = \phi + \nabla \phi \cdot \Delta \hat{s} \]  \hfill (5) 

where \( \phi_x \), \( \phi \), \( \nabla \phi \) and \( \Delta \hat{s} \) are the face value, cell-centered value, gradient from the upstream cell and displacement vector from the centroid of the upstream cell to the face centroid, respectively. The Taylor series expansion about the cell centroid requires the application of the divergence theorem to solve the \( \nabla \phi \) terms for each cell as follows

\[ \nabla \phi = \frac{1}{V} \sum_j \bar{\phi}_j \cdot \hat{A} \]  \hfill (6) 

where \( \bar{\phi}_j \) is the average value of \( \phi_j \) for the two adjacent cells. The SIMPLE pressure-velocity coupling scheme was employed. This scheme relates the pressure at a CV face to the velocity at the same face by ensuring mass flux is conserved across a given cell.

**Linearization**

In order to calculate the unknown quantities, a set of coupled non-linear equations must be solved. In order to solve the coupled equations, they must first be linearized. This set of linearized algebraic equations makes up a sparse coefficient matrix which is then solved using a Gauss-Seidel solver combined with an algebraic multigrid method (Fluent) for all scalar equations. A segregated implicit solver scheme was employed which solves each variable for each control volume sequentially using existing values and unknown neighboring values. As the equations are non-linear, several iterations are required in order to achieve convergence.

**Under-Relaxation Factors**

The discretization schemes rely on under-relaxation factors to ensure smooth convergence of all non-linear equations; this is done by controlling the total change of variable \( \phi \) from one iteration to the next using the following equation,

\[ \phi = \phi_{old} + \alpha \Delta \phi \]  \hfill (7) 

where the new value \( \phi \) is obtained from the previous value \( \phi_{old} \) and a percentage \( \alpha \) of the calculated change \( \Delta \phi \) between the old and new value. The multiplying factor \( \alpha \) is called the under-relaxation factor. The default values were used for all under-relaxation factors (Fluent).

**Residuals**

After each iteration, the values of the conserved quantities were calculated using a scaled residual.
$R^\phi = \sum_{\text{cells}} \left( \sum_{\text{nb}} a_{nb} \phi_{nb} + b - a_{P} \phi_{P} \right) \sum_{\text{cells}} a_{P} \phi_{P}$ \hspace{1cm} (8)

where $a_P$ and $a_{nb}$ are the linearized coefficients for $\phi$ and $\phi_{nb}$ of equation (9). The subscript “nb” refers to the neighbouring cells.

$$a_{P} \phi = \sum_{\text{nb}} a_{nb} \phi_{nb} + b \hspace{1cm} (9)$$

**FLUID DYNAMIC VALIDATION**

The initial model development began with the modeling of laminar flow between parallel plates without including the effects of radiation. Once the laminar validation was complete, turbulent flow was then considered. In order to validate the model, a Nusselt (Nu) number analysis was used to help determine the accuracy of the various mesh densities. Using the data obtained from Fluent, the Nu values were calculated using two different equations,

$$Nu_A = \frac{\partial T}{\partial X_{\text{wall}}} \frac{D_h}{T_{\text{wall}} - T_{\text{mean}}} \hspace{1cm} (10)$$

and

$$Nu_b = \frac{q'^{\text{wall}} D_h}{(T_{\text{wall}} - T_{\text{mean}}) \kappa_f} \hspace{1cm} (11)$$

where $T_{\text{mean}}$ is the mass-averaged local mean temperature between both walls given by

$$T_{\text{mean}} = \frac{\int \rho v C_v T dA_v}{m C_v} \hspace{1cm} (12)$$

and $D_h$ is the hydraulic diameter calculated as

$$D_h = \frac{4 A}{\phi} \hspace{1cm} (13)$$

For the case of two parallel plates with infinite length, the hydraulic diameter is $D_h = 2W$ where $W$ is the cavity width of 92 mm. Calculating two Nu numbers helped provide insight into the mesh performance, particularly in the region immediately adjacent to the wall where the velocity, temperature and shear gradients are most pronounced. It is expected that for a specified heat flux at the wall, the value obtained from Equation (11) should be more accurate as it is directly proportional to $q'^{\text{wall}}$. The value for $Nu_b$ is a function of the temperature gradient at the wall, the accuracy of $Nu_b$ was dependent on the mesh density throughout the control volume (and particularly near the wall) as well as the accuracy of Fluent’s near-wall treatment methods. The value of $Nu_b$ in relation to $Nu_A$ was therefore used as a performance indicator for the different mesh configurations, helping quantify the accuracy of the near-wall gradients for each of the different mesh densities.

Within Fluent, seven different models are available for modeling turbulence. Of the seven models, the three most suitable were: standard $k$-$\epsilon$, standard $k$-$\omega$ and the Shear-Stress Transport (SST) $k$-$\omega$ model. The standard $k$-$\epsilon$ model, which is one of the most widely used turbulence models, is applicable to wall-bounded flows while the SST model is a mix of both the standard $k$-$\epsilon$ and $k$-$\omega$ model; in the near-wall regions, the SST model makes use of the standard $k$-$\omega$ model while away from the wall, it relies on the standard $k$-$\epsilon$ model. A blending function is used to activate the appropriate model in the appropriate region and provide a mixture of the two otherwise. According to the Fluent literature, all three models, namely the standard $k$-$\epsilon$, the standard $k$-$\omega$ and the SST $k$-$\omega$ models, are appropriate for modeling the given flow.

The turbulent boundary layer, which can be modeled as three sublayers (viscous sublayer, buffer layer and fully-turbulent layer) as shown in Figure 4, is modeled in Fluent using either standard wall functions (SWF) or enhanced wall treatments (EWT).

The SWF resolves the viscous sublayer and the buffer layer mathematically without requiring any mesh in those regions using semi-empirical correlation data. The EWT requires a very particular mesh near the wall in order to resolve the boundary layer regions. A schematic highlighting the distinctions between both methods can be seen in Figure 5. Using SWF, the recommended (Fluent) location of the first cell centroid is $y^+ \approx 30$ where
\[ y^+ = \frac{\rho u_L y}{\mu} \]  
\[ u_c = \sqrt{\frac{\tau_w}{\rho}} \]  

Figure 5. Comparison between SWF and EWT.

In contrast to the SWF, Fluent recommends \( y^+ \approx 1 \) for EWT in order to resolve the near-wall gradients.

Results and Discussion
Using all three models (standard k-\( \varepsilon \), standard k-\( \omega \) and SST k-\( \omega \)) and both wall treatments (SWF and EWT), a fully developed Nu number analysis was undertaken to decide which wall treatment and which model would be best suited for modeling the AFW section of the BIPV/T system. Using the recommended mesh spacing schemes, the fully developed Nu numbers were calculated for turbulent flow. From these results, it was decided that the SST k-\( \omega \) model with EWT produced the best results and that this model would be used for further model refinements. A developing Nu number analysis was then performed with the SST k-\( \omega \) with EWT model. The results were compared to published data by two parties, namely Azer et al. (1960) and Hatton et al. (1963). The mesh that performed the best was one with a uniform mesh spacing throughout the cavity as identified by Case 8 in Figure 6. In Figure 6, the published data by Azer et al. (1960) and Hatton et al. (1963) is represented by A and C as well as H and Q, respectively.

THERMODYNAMICS VALIDATION
Having validated a mesh for forced convection between parallel plates, the focus was then placed on adding radiation heat transfer to the model. Radiation heat transfer can be split into two steps: short wave and long wave analysis. In order to perform the short wave analysis, software used for window analysis called VISION (1996) was employed.

![Developing Nusselt Number](Image)

Figure 6. Comparison between SWF and EWT.
VISION

Using VISION, an optical analysis was performed. Based on the given indoor and outdoor conditions as well as the optical and thermal properties of each layer, the absorbed energy at each layer was calculated. These values could then be added to the Fluent model as volumetric source terms. Fluent was then used to perform a heat transfer analysis, including conduction, natural convection and long wave radiation exchange. For the purpose of the validation, the temperature at each layer provided by VISION was compared to the temperature profiles generated by Fluent for buoyancy-induced air flow.

Radiation Model

Within Fluent, five different models exist to solve radiation heat transfer. For problems involving semi-transparent walls, such as shading devices, the Discrete Ordinates (DO) model must be used. Using the DO model, the radiative transfer equation is solved like any other transport equation where the radiation intensity is transported from one solid boundary layer to another. Throughout the modeling, the air was assumed to be a non-participating medium. The discretization of the medium is achieved using user-specified solid angles which are subdivided further by pixels. The energy contained within each pixel is calculated as the sum of the incoming energy minus the sum of the outgoing energy. With the exception of the blind, all walls were treated as opaque to long wave radiation.

Results and Discussion

In order to validate the numerical model, four different fenestration systems were modelled in VISION: single-glazing (case 1), double-glazing (case 2), triple-glazing (case 3) and double-glazing with between-the-panes venetian blind (case 4). The temperature values generated at each surface in VISION were compared to the temperature values in Fluent. The following table highlights the results for two of the validation exercises, namely the case with triple-glazings and double-glazing with between-the-panes venetian blind.

As was the case for all four validation scenarios, Fluent was able to accurately calculate all surface temperatures.

COMBINED CONVECTION-RADIATION VALIDATION

Once the numerical model had been validated for laminar and turbulent convection as well as radiation, the model was then validated for turbulent flow with radiation heat transfer. This was achieved using experimental data obtained from the Solar Lab at Concordia University.

<table>
<thead>
<tr>
<th>Glazing Identification</th>
<th>Temperature [k]</th>
</tr>
</thead>
<tbody>
<tr>
<td>Descripti</td>
<td>Glazing</td>
</tr>
<tr>
<td>Case 3 with an irradiance of G = 783 W/m²</td>
<td>A 1</td>
</tr>
<tr>
<td></td>
<td>2</td>
</tr>
<tr>
<td></td>
<td>B 3</td>
</tr>
<tr>
<td></td>
<td>4</td>
</tr>
<tr>
<td></td>
<td>C 5</td>
</tr>
<tr>
<td></td>
<td>6</td>
</tr>
<tr>
<td>Case 4 with an irradiance of G = 783 W/m²</td>
<td>A 1</td>
</tr>
<tr>
<td></td>
<td>2</td>
</tr>
<tr>
<td></td>
<td>B 3</td>
</tr>
<tr>
<td></td>
<td>4</td>
</tr>
<tr>
<td></td>
<td>C 5</td>
</tr>
<tr>
<td></td>
<td>6</td>
</tr>
</tbody>
</table>

Particle image velocimetry (PIV) was used to obtain a velocity profile in the cavity on March 13th, 2006 at 1:31 p.m. At this time, temperature data was also collected using T-type thermocouples that were placed along the insides of both glazings.

**Results and Discussion**

The velocity profile obtained from the PIV is compared to that obtained from Fluent in Figure 7. It can be seen that the numerical results are well aligned with the experimental results. The one deviation is on the right side of the blind where a pressure probe was affecting the flow causing a sudden drop in the experimental velocity profile. This probe caused a sudden acceleration of the flow on either side; for this reason, the shape of the experimental velocity profile is not as flat across the top as one would expect with turbulent flow.

The temperature data was also compared along the inner edges of both the outside and inside glazings. The results can be seen in Figure 8. The abbreviations lwls and rwls located in the legend stands for left wall left side and right wall right side, referring to the inner layers of the right and left glazings, respectively. As was the case for the velocity data, the temperature data matches well with the experimental data.
Comparison between Experimental and Numerical Velocity Profiles

Figure 7. Velocity comparison between data obtained from Fluent and experiment.

Airflow Window Temperatures

Figure 8. Temperature comparison between data obtained from Fluent and experiment

CONCLUSION

Throughout this paper, the validation process for the development of a numerical model for an AFW has been presented. A three stage validation process was applied whereby a convection validation was followed by a radiation validation before a final combined convection/radiation validation was performed. For each of the three stages, the validation was shown to be in excellent agreement with other data sources, including analytical, numerical and experimental work. In order to model forced convection in the turbulent regime, the SST k-ω model was shown to produce the best results with a uniform grid throughout the flow domain. For the radiation validation, VISION was required to perform the optical analysis needed to generate the volumetric source terms at each layer. Fluent was then validated by comparing the temperatures at each layer to those calculated by VISION. Finally, velocity and temperature profiles were obtained from Fluent and compared to experimental data collected from the Solar Lab at Concordia University. All of the data generated by Fluent was in excellent agreement with the experimental results. In the end, a fully validated model exists which can be used in the future to run optimization exercises of the BIPV/T system.
REFERENCES


FLUENT 6.2. A computational fluid dynamics software used to model fluid flow, heat and mass transfer.


