CONDENSATION RISK ASSESSMENT OF WINDOW-WALL FACADES UNDER THE EFFECT OF VARIOUS HEATING SYSTEMS

D. Yan and R. Mora

ABSTRACT

In cold climates, windows are typically the coldest surfaces in the interior of a wall assembly, causing thermal discomfort, heat losses, and moisture condensation. This paper investigates the interactions between various heating systems and window-wall systems through convection and radiation heat exchanges, and their effects on surface condensation. Three common heating systems for multi-unit residential buildings were evaluated: electric baseboard, radiant floor and forced air system. Each heating system provides vastly different indoor conditions due to differences in thermal stratification, room air distribution and location of the heater. These differences have direct impact on the risk of condensation in a window. In this project, two typical window wall details were studied using THERM (LBNL 2013) finite element modeling software under boundary conditions dependent on the heating system studied. Convective heat transfer coefficients were drawn from the literature in an attempt to represent the air flow and radiation induced by each heating system. Computational Fluid Dynamics (CFD) (Autodesk 2013) modeling was used in an attempt to model the boundary conditions more accurately for convective systems. The analysis showed that THERM model could provide results that were consistent with those of the CFD models; however, these two models varied by as much as 30%. In general, a forced air system was considerably more susceptible to window condensation risk which varies depending on the inlet location, as well as supply air speed and temperature. The radiant floor system also results in significant condensation risk when the indoor relative humidity is above 55%. A second phase of this project will seek to calibrate the models through monitoring and measurements, explore improve window-wall constructive solutions to minimize the risk of condensation, and conduct sensitivity analyses to boundary conditions.

1. INTRODUCTION

The goal of this paper is to investigate how boundary conditions that are created by different heating systems in a typical multi-unit residential building (MURB) affect the risk of condensation in window wall systems. The three most common heating systems that are used in high rise residential building were selected: electric baseboard, radiant floor and forced air system. Each heating system provides vastly different indoor boundary conditions due to differences in thermal stratification, air distribution in the room and location of the heater.

In order to investigate the effects of these heating systems, it is important to understand each of the heat transfer mechanisms involved, i.e. conduction, convection and radiation. While conduction and radiation can be modeled accurately via the use of heat transfer simulation software, it is not the case for convection because convective heat transfer is highly sensitive to buoyant and mechanically induced air movements as discussed by Beausoleil-Morrison (2002).

Currently, there are two available methods to model convection coefficients in building simulation: 1) empirical coefficients obtained from laboratory experiments, 2) computational fluid dynamics (CFD) simulation. In this project, the two methods were explored and were used to model the selected window wall details. The software THERM (LBNL 2013) and Simulation CFD (Autodesk 2013) were selected to simulate the condensation risk of these typical glazing units with different heating systems. THERM was used to model the two-dimensional (2D) heat transfer through envelope details, including cavities, using...
the finite element method (FEM) with boundary conditions given by convective and radiation heat transfer coefficients. CFD was used to predict the air flow patterns induced by the heating systems, under the boundary conditions provided by the surface temperatures and the supply air provided by a forced air system. Room air-flow obstacles and people can also be included in the CFD simulations.

The questions to be answered by this project were the following. How significant is the impact of room air flow on condensation risk in window wall systems? Are empirical film coefficients sufficient for predicting condensation risk of window wall units? What are the quantitative differences between each of the heating systems on condensation risk? The project designed a methodology in an attempt to better understand and predict these physical phenomena and will hopefully guide further efforts to better characterize the effect of different heating systems in window condensation risk analysis.

2. RESEARCH METHODOLOGY

The methodology followed in this project is described in Figure 1 below. As a point of departure, the boundary conditions were investigated from the literature for the window detail (THERM) and the room air flow models (CFD). In parallel, relevant window-wall details were selected from the local industry. Finally, 2D heat transfer models for the window detail were constructed in THERM, and CFD room air flow models were created with representative boundary conditions for each heating system. The models were steady-state under the typical winter conditions of Vancouver.

FIGURE 1: METHODOLOGY FLOW CHART OF THE PROJECT
2.1. THERM MODELLING

Two common multi-unit residential building window wall details were selected for the project: window wall assembly with bypass spandrel glass panel, and extended slab edge with window wall assembly. THERM models were built to determine the indoor surface temperatures at the critical window-wall details for condensation. The models were built under the guidelines by National Fenestration Rating Council (NFRC) (2011). 0°C was selected as the outdoor boundary condition to reflect a typical winter temperature. The exterior film coefficient was set as 29 W/m²K, using data from ASHRAE (2009). The models implemented in THERM of the bypass window wall system (Figure 2, left) and the extended slab systems (Figure 2, right) were identified below.

![Image of THERM models on bypass detail and extended slab edge detail]

In THERM models, indoor boundary conditions are described by heat transfer coefficients of window assembly surfaces and a constant value is typically used as discussed by Arasteh (2003). Reference models use a fixed, combined surface heat-transfer coefficient for radiation and convection that is based on NFRC guidelines (2011).

2.2. HEATING SYSTEM MODELS IN THERM

The effect of each heating system was modeled using corresponding convective and radiation heat transfer coefficients. The convective heat transfer coefficients used in THERM were drawn from the literature, as illustrated in Table 1.

<table>
<thead>
<tr>
<th>Detail</th>
<th>Heating system</th>
<th>Source</th>
<th>( h_c )</th>
<th>Reference temperature</th>
</tr>
</thead>
<tbody>
<tr>
<td>Slab Edge</td>
<td>Reference</td>
<td>ASHRAE (2009)</td>
<td>6.8 (Fixed)</td>
<td>Average room air temperature</td>
</tr>
<tr>
<td>Slab Edge</td>
<td>Electric baseboard</td>
<td>Khalifa, et al (1990)</td>
<td>( h_c = 8.07\Delta T + 0.11 )</td>
<td>Average room air temperature</td>
</tr>
<tr>
<td>Slab Edge</td>
<td>Radiant Floor</td>
<td>Khalifa, et al (1990)</td>
<td>( h_c = 7.61\Delta T + 0.06 )</td>
<td>Average room air temperature</td>
</tr>
<tr>
<td>Slab Edge</td>
<td>Forced Air</td>
<td>Goldstein, et al (2010)</td>
<td>( h_c = 0.103(V/L)0.8 )</td>
<td>Supply air temperature</td>
</tr>
<tr>
<td>Bypass</td>
<td>Reference</td>
<td>ASHRAE (2009)</td>
<td>6.8 (Fixed)</td>
<td>Average room air temperature</td>
</tr>
<tr>
<td>Bypass</td>
<td>Electric baseboard</td>
<td>Khalifa, et al (1990)</td>
<td>( h_c = 8.07\Delta T + 0.11 )</td>
<td>Average room air temperature</td>
</tr>
<tr>
<td>Bypass</td>
<td>Radiant Floor</td>
<td>Khalifa, et al (1990)</td>
<td>( h_c = 7.61\Delta T + 0.06 )</td>
<td>Average room air temperature</td>
</tr>
<tr>
<td>Bypass</td>
<td>Forced Air</td>
<td>Goldstein, et al (2010)</td>
<td>( h_c = 0.103(V/L)0.8 )</td>
<td>Supply air temperature</td>
</tr>
</tbody>
</table>

TABLE 1: LIST OF CONVECTIVE HEAT TRANSFER COEFFICIENTS USED IN THERM
The radiation component was modeled explicitly through the use of a view-factor-based radiation model. The conductive components of the models are simulated via the use of a heat source. For electric baseboard models, a radiating surface with a fixed temperature was used (Figure 3, left). The radiating surfaces were basically the surfaces adjacent to baseboard heater which followed the exact size of a typical electric baseboard heater. For the radiant floor model, floor pipes and surface with set temperatures were used (Figure 3, right). The input parameters are selected via the use of manufacturer data as discussed by Cadet [8] and design guide as discussed by Bean (2006).

FIGURE 3: THERM ELECTRIC BASEBOARD MODEL AND FIGURE 5 THERM RADIANT FLOOR MODEL

2.3. COMPUTATIONAL FLUID DYNAMICS (CFD) MODELING

Two dimensional (2D) Computational Fluid Dynamics (CFD) models were created, using CFD simulation software (Autodesk 2013), which couples room air flow and window-wall heat transfer. A total of eight CFD models were built: one for electric baseboard and radiant floor, two for forced air system with variant on supply inlet location, and a set of these for each window wall details: bypass detail and extended slab edge detail. Due to the capability of simulating fluid flow in CFD model, no convective coefficients were required in the boundary condition. For the radiation portion of the heat transfer, the radiation option was enabled in CFD model. The conductive components were simulated via the use of heat sources. For the electric baseboard models, a radiating surface was implemented at where the heater is located (Figure 4, left). For the radiant floor model, floor pipes and floor surfaces with set temperature were used (Figure 4, right). For the boundary conditions, temperature of the radiant floor and pipe surfaces were set at a fixed value.

FIGURE 4: CFD ELECTRIC BASEBOARD (LEFT) AND RADIANT FLOOR (RIGHT) MODEL
Two configurations for location of the supply inlet were implemented in the forced air models (Figure 5). As discussed by Beausoleil-Morrison (2011), the location of supply inlet had a huge impact on the performance of the heating system.

FIGURE 5: CFD FORCED AIR SYSTEM MODEL WITH CENTRAL AND END SUPPLY INLET & BYPASS DETAIL

2.4. DATA REFERENCES

To plot a graph of surface temperatures against locations on the glazing unit, an origin point was set at the location where the window glass and the frame met at the sill section, which is called the “Sight Line” in Figure 6. Positive sight line distance values are for points on the window frame below the sight line and vice versa. The Y-axis represented surface temperature in °C. Two locations on the glazing units were expected to be the coldest were named 1st and 2nd cold corners as noted in figure 6.

FIGURE 6: DATA REFERENCES FOR THE THERM MODEL

3. ANALYSIS AND RESULTS

Due to the uncertainties in the inputs of the heating systems, ranges of parameters were used in the models to demonstrate their sensitivities and effects on the result. The data generated in this sensitivity analysis were used to improve the accuracy of the comparison between each heating system. The 1st and 2nd cold corner of each model were assessed with consideration of the possible margin of difference (+/-).
3.1 THERM MODEL ANALYSIS

Surface temperatures data were generated at each reference and for each of the heating system models and then the data were plotted under one graph. The dew point threshold of 21°C and both 50% and 60% relative humidity were plotted to assess condensation risk of each model. Data from the sensitivity analysis were used to demonstrate the possible result range when various input parameters were used on the heating systems models. Results showed that the general patterns of surface temperature in all models were consistent for both the extended slab edge and the bypass detail. The electric baseboard models in general had the highest surface temperature out of all the models for both details, while the forced air model had the lowest general surface temperature (Figure 7).

![Figure 7: Surface Temperature of Bypass detail using THERM](image)

In the bypass models, the location of the lowest surface temperature was at the 1st cold corner for electric baseboard one and at 2nd cold corner for the rest (table 3). The surface temperatures at the cold corners for electric baseboard model were above both dew point temperature thresholds. For the other three models, their surface temperatures at the 2nd cold corner were above the 21°C 50% RH dew point temperature but below the 60% RH one.

<table>
<thead>
<tr>
<th>Type of Detail</th>
<th>Type of Model</th>
<th>ΔT at 1st cold corner</th>
<th>ΔT at 2nd cold corner</th>
</tr>
</thead>
<tbody>
<tr>
<td>Bypass</td>
<td>REFERENCE</td>
<td>11.7</td>
<td>11.6</td>
</tr>
<tr>
<td>Bypass</td>
<td>Electric Baseboard</td>
<td>14.6(+/- 0.6)</td>
<td>17.8(+/- 1.9)</td>
</tr>
<tr>
<td>Bypass</td>
<td>Radiant Floor</td>
<td>11.3(+/- 0.1)</td>
<td>10.4(+/- 0.1)</td>
</tr>
<tr>
<td>Bypass</td>
<td>Forced Air</td>
<td>9.6(+/- 1.7)</td>
<td>9.3(+/- 1.5)</td>
</tr>
</tbody>
</table>

TABLE 3: SURFACE TEMPERATURES AT COLD CORNERS FOR BYPASS DETAILS - THERM

3.2 CFD MODEL ANALYSIS

The general patterns of surface temperature in all models were consistent for both the extended slab edge and the bypass detail. The electric baseboard models in general had the highest surface temperatures out of all the models for both details. The order followed with radiant floor model, forced air with mid inlet model and forced air with end inlet model, which had the lowest general surface temperatures.
In all of the bypass models, the location of the lowest surface temperature was at the 1st cold corner (Table 4); only the electric baseboard model was above both the 21°C 50% and 60% RH dew point temperature at this location. All the other models were above the 50% RH threshold and below the 60% RH one. The surface temperature of the mid inlet model was slightly higher than the end inlet one, by 0.9 °C.

<table>
<thead>
<tr>
<th>Type of Detail</th>
<th>Type of Model</th>
<th>ΔT at 1st cold corner</th>
<th>ΔT at 2nd cold corner</th>
</tr>
</thead>
<tbody>
<tr>
<td>Bypass</td>
<td>Electric Baseboard</td>
<td>17.6</td>
<td>20.7</td>
</tr>
<tr>
<td>Bypass</td>
<td>Radiant Floor</td>
<td>12.3</td>
<td>12.4</td>
</tr>
<tr>
<td>Bypass</td>
<td>Forced Air mid inlet</td>
<td>10.3</td>
<td>11.3</td>
</tr>
<tr>
<td>Bypass</td>
<td>Forced Air end inlet</td>
<td>9.4</td>
<td>11.6</td>
</tr>
</tbody>
</table>

**TABLE 4: SURFACE TEMPERATURE OF COLD CORNERS FOR BYPASS DETAILS - CFD**

In general, models for the bypass detail had higher surface temperatures than the ones for the extended slab edge detail. The surface temperature was as low as 5.7 °C in extended slab edge model, while it only reached 9.4 °C in bypass model.

### 3.3 MODELING TOOLS COMPARISON

This section demonstrates quantitatively the differences in the results from the THERM and CFD simulations. For the bypass detail and the electric baseboard models, the lowest surface temperature of the THERM model was 3°C lower than the CFD one (Figure 9). For the radiant floor models, the lowest surface temperature of the THERM one was 1.9°C lower than the CFD one. For the forced air models, the lowest surface temperature of the THERM one was 0.1°C higher than the CFD one.

The THERM models varied by as much as 30% in surface temperature compared to CFD modeling. In general, the THERM model overestimated the surface temperature of the cold corners for radiant floor and forced air system; on the other hand, it underestimated the result of electric baseboard model compared to CFD modeling.
3.4 **CFD: AIR FLOW PATTERNS AND TEMPERATURE DISTRIBUTIONS**

This section illustrates the air distribution and thermal stratification of the window sill sections and the room for each heating system model for the bypass detail. The colour differences reflect temperature differences. The arrows represent the direction of air flow and the size of arrow represents the speed.

### 3.4.1 ELECTRIC BASEBOARD MODELS

The window sill section shows that there was an upward convective heat flow due to the heat sources on the radiating surfaces, which simulated the effect of the baseboard heater (Figure 10). Overall, the electric baseboard heater was able to distribute heat evenly at the center of the room (Figure 11).
3.4.2 RADIANT FLOOR

The window sill section shows a downdraft flow towards the windows sill (Figure 12). The room air shows that there is little thermal stratification (Figure 13). Overall, the radiant floor system was able to distribute heat more uniformly within the room, except for the cold corner at the window sill.
3.4.3 FORCED AIR WITH CENTRAL INLET

The window sill section shows an air flow carried by upward momentum traveling along the fenestration unit and towards the window (Figure 14). However, this air flow does not provide sufficient heat for the cold corners, as revealed by the temperature gradient.

![Figure 14: Air Flow Patterns - Window Sill of Forced Air Mid Inlet Model](image1)

FIGURE 14: AIR FLOW PATTERNS - WINDOW SILL OF FORCED AIR MID INLET MODEL

The room air show flow patterns that the forced air system creates multiple convective air loops within the room (Figure 15). The forced air system does not distribute air as evenly as the radiant floor system and recirculation zones appear at cold corners. There appears to be a cold corner at the end of the room with a small localized convective air loop.

![Figure 15: Air Flow Patterns - Room of Forced Air Mid Inlet Model](image2)

FIGURE 15: AIR FLOW PATTERNS – ROOM OF FORCED AIR MID INLET MODEL

3.4.4 FORCED AIR WITH INLET AT THE END OF ROOM

The window sill section shows that there is a downdraft flow along the window toward both of the cold corners (Figure 16). It is apparent that the cold corners are at high risks of condensation. The room air shows that the forced air system creates multiple convective air loops with in the room (Figure 17). It shows that the forced air system does not distribute air evenly at all. The area close to the exterior wall is not heated up sufficiently.
4. DISCUSSION AND RESEARCH FINDINGS

In typical THERM analyses, the simplified way to model the interior boundary conditions, which was represented by the REFERENCE model, did not accurately represent any of the indoor conditions that were under the effect of any of the heating systems. The REFERENCE model tended to underestimate the temperatures compared to those from electric baseboard systems and overestimate those from both the radiant and forced air systems.

Both the THERM and CFD analyses revealed that the electric baseboard model in general, is least susceptible to condensation risk under the configurations of the window details and exterior winter condition that were employed in this project. The radiant floor system results in significant condensation risk when the indoor relative humidity is above 55%. The forced air model was most susceptible to condensation risk out of all the heating systems. In the CFD analysis for the forced air system; the result showed that the forced air model with central inlet performed slightly better than the one with end inlet.
The modeling tool analysis showed that the result for radiant floor model was consistent. For the electric baseboard model, the surface temperature from the CFD model was 3°C to 5°C higher than the THERM model. This could be explained by the difference between the way radiating surfaces were used to simulate the effect of the electric baseboard heater and how the actual heater worked. For the forced air model, the surface temperature from CFD model was roughly 2°C lower than THERM model. This could be explained by the difference in configuration of the location of the inlet. The analysis showed that THERM model could provide results which were consistent with those of the CFD models; however, these two models varied by as much as 30%.

As expected, the bypass models performed better than the extended slab edge details in surface condensation resistance. This was due to the fact that the extended slab edge provided a thermal bridge for the window assembly.

5. CONCLUSION AND FURTHER WORK

The research findings confirmed the hypothesis that the window condensation risk is affected by the heating system. The major finding in this project is that the typical modeling method of using a fixed interior boundary coefficient is not sufficient for describing a realistic indoor boundary condition, in which a heating system is present in a room under winter conditions in northern coastal climate. Each heating system provides vastly different indoor conditions due to differences in thermal stratification, air distribution in the room and location of the heater. These differences have direct impacts on window performance and affect the risk of condensation. The research further confirms that thermal bridging in the studied detail increase the chance of surface condensation in a fenestration system.

Based on the research findings, it appears that an accurate implementation of indoor boundary conditions is required to accurately assess condensation risk of window wall assemblies with typical heating systems. In addition, CFD simulation provided meaningful insights into how air flow affects the condensation risk in window assemblies. Future work includes developing three dimensional (3D) CFD models to evaluate the effects of 3D supply forced-air flows at the room-window corners. Other relevant factors to be considered are: the presence of furniture and blinds in reducing convection and radiation heat transfer. A second phase of this project will seek to calibrate the models through monitoring and measurements, explore improve window-wall constructive solutions to minimize the risk of condensation, and conduct sensitivity analyses to boundary conditions.

REFERENCES

ASHRAE (2009). ASHRAE Handbook of Fundamentals, American Society for Heating Refrigerating and Air Conditioning Engineers, Atlanta, USA.