MODELLING MIXED CONVECTION HEAT TRANSFER AT INTERNAL BUILDING SURFACES

Ian Beausoleil-Morrison[1]

[1]CANMET Energy Technology Centre, Natural Resources Canada, Ottawa Canada
PhD Student, Energy Simulation Research Unit, University of Strathclyde, Glasgow U.K.
(ibeausol@nrcan.gc.ca)

ABSTRACT

The treatment of convective heat transfer at internal building surfaces has a significant impact on the simulation of heat and air flow. Accurate approaches for the range of flow regimes experienced within buildings (buoyant flow adjacent to walls, buoyant plumes rising from radiators, fan-driven flows, etc.) are required, as is the ability to select an appropriate method for the case at hand and to adapt modelling to changes in the flow.

A new approach—drawing upon previously published methods—has been developed for modelling mixed convection within mechanically ventilated rooms. It is applicable for rooms ventilated with ceiling mounted diffusers and is appropriate for both heating and cooling. ESP-r simulations performed with the mixed flow model indicate that the prediction of heating and cooling loads is highly sensitive to the treatment of surface convection and that significant errors can result if an inappropriate model is employed. The results also reveal that the choice of convection algorithm can influence design decisions drawn from a simulation-based analysis.

INTRODUCTION

Building simulation (BSim) has evolved considerably from its origins over three decades ago. The early BSim tools were strictly thermal models, used to estimate building energy consumption and peak heating and cooling loads. Although these early tools considered the thermal impact of user-prescribed infiltration and ventilation rates, the flow of air was not simulated, nor was the interdependency of heat and air flow considered.

The scope of BSim has widened considerably in recent years to consider physical processes which are not strictly thermal. As well, there has been a trend towards integrated modelling:

- The simulation of building loads and plant equipment has been integrated.
- Macro-scale air flow models have been incorporated to couple the simulation of heat and air flow and to analyze pollutant dispersion within buildings.
- Illumination models have been coupled to enable the assessment of visual comfort and to consider the interactions between thermal and visual performance (e.g. occupants closing blinds in response to glare).

In addition to widening the scope of BSim and integrating modelling methods, advanced models have been developed for some of the important heat transfer mechanisms:

- Conduction transfer function and finite-difference techniques are well-developed and widely utilized for modelling transient heat transfer through the building fabric.
- Ray-tracing approaches for view-factor calculation in conjunction with radiosity models are often used for inter-surface radiation exchange.
- Detailed ground-contact models have been created to consider the impact of time-varying ground temperatures and the transient heat storage of the surrounding soil.

BSim will continue to evolve towards more integrated and more highly resolved modelling approaches, driven by the need to address the complex nature of real-world design and analysis problems. One area where further refinement is necessary is the treatment of convective heat transfer at internal building surfaces (e.g. walls, windows). This is the topic of the current paper and a key element in a research effort aimed at advancing the integrated modelling of heat and air flow in buildings.

This paper outlines a number of issues pertaining to the modelling of internal surface convection, placing the significance of this heat flow path in context. A new procedure for modelling mixed-flow convection is then described. Following this, a series of ESP-r (ESRU 1997) simulation results are presented to demonstrate the application of the new approach and to illustrate the sensitivity of BSim results to the treatment of internal convection.

IMPORTANCE OF CONVECTION

Treatment of Surface Convection in BSim

A number of BSim programs treat surface convection as an explicit heat flow path, although many combine convection with inter-surface radiation, modelling the two processes with a "film" coefficient and solving for some fictitious "operational" temperature.

For those programs that do treat internal surface convection explicitly, most employ the so-called *well-stirred* assumption. This treats the room air as uniform and characterizes surface convection by a convection coefficient (h_c) and the temperature difference between the room air (T_{air}) and the solid surface ($T_{surface}$, also assumed to be of uniform temperature):

$$q''_{conv} = h_c \cdot (T_{air} - T_{surface}) \quad (1)$$

where q''_{conv} is the convective heat flux from the air to the surface.

Some BSim programs employ time-invariant h_c values (either user-prescribed or "hard-wired" in the program's source code), although many recalculate h_c for each surface each time step, using some user-selected or fixed equation. Many h_c equations are in use, some are particular to buildings and are appropriate for specific flow regimes, while others are general relations from the heat transfer literature.

Relevance to Thermal Modelling

Given these simplified and varied approaches it is not surprising that the recently completed IEA BESTEST project (Judkoff and Neymark 1995)—the objective of which was to systematically test and diagnose sources of disagreement between BSim programs—identified the modelling of surface convection to be one of the primary causes of disagreement between programs.

Numerous researchers have examined the sensitivity of BSim thermal predictions to the modelling of internal convection (e.g. Waters 1980; Irving 1982; Spitler et al 1991; Clarke 1991; Fisher and Pedersen 1997). Their work has demonstrated that predictions of energy demand and consumption can be strongly influenced by the choice (made by program developer or user) of h_c algorithm. Differences of 20-40% in energy predictions were noted by some of these authors.

More importantly, the predicted benefits from design measures were, in some cases, found to be sensitive to the approach used to model internal surface convection. As a result, the choice of h_c algorithm could affect the design decisions drawn from a simulation-based analysis. These observations alone provide ample motivation to improve the modelling of internal surface convection within BSim.

Relevance to Air Flow Modelling

The significance of internal convection modelling is not limited to thermal simulations, however, but is also of relevance in modelling indoor air motion.

Computational Fluid Dynamics (CFD) has been widely and successfully applied in the prediction of room air motion (e.g. Jones and Whittle 1992). However, accuracy is—as with all modelling techniques—highly sensitive to the boundary conditions supplied (assumed) by the user (e.g. Awbi 1998).

The application of boundary conditions in BSim is relatively straightforward. The model boundary is (typically) placed at the exterior of the building fabric: boundary conditions can be established in the form of exterior conditions—dry-bulb temperature, wind velocity, etc.—drawn from an appropriate weather-data file. However in modelling room air flow with CFD, the model boundary is located *within* the building: the user must supply boundary conditions in the form of internal wall conditions (surface temperatures or heat flow) and air flows entering/leaving the room. The fundamental dilemma is clear. A room does not exist in isolation: wall temperatures and air flows through openings are dynamic and dependent on the external weather conditions, states prevailing throughout the rest of the building, and the operation of plant equipment, these in turn depending on conditions within the room.

CFD researchers have begun to address this issue by integrating dynamic fabric models and inter-surface radiation models into CFD codes (e.g. Holmes et al 1990). This allows room air flow to be calculated by prescribing boundary conditions external to the building or in adjoining spaces, rather than within the room.

Integration of CFD into BSim

Negrão (see Clarke et al, 1995) extended this concept by integrating a CFD code into ESP-r, the two models operating in tandem, "handshaking" on a time-step basis. A thermal and (optionally) a network air flow representation of the whole building and plant is established in the BSim program while a CFD model is created for a single room. BSim establishes the boundary conditions for the CFD model. Once the CFD solution converges, it passes thermal or air flow results to the BSim model, which uses the data to calculate the surface temperatures, energy flows, and air flows throughout the building. This process is repeated each time step. The reader is referred to Clarke and Beausoleil-Morrison (1997) for an overview of the "handshaking" mechanisms.

The power of the BSim-CFD integrated modelling approach is clear. BSim has the potential to supply realistic time-varying boundary conditions for CFD, while CFD has the potential to predict the details of flow and temperature fields within particular zones, thus enabling flow visualisation, studies on pollutant dispersion, and thermal comfort assessments.

Success of the BSim-CFD approach, however, is critically dependent upon the treatment of the physics at the model boundaries, the locations at which the BSim and CFD systems interact. Consequently, any errors in the modelling of surface convection will be propagated (perhaps amplified): if BSim supplies inaccurate boundary conditions for CFD, CFD will calculate an incorrect temperature and flow field for the zone; the erroneous results passed from CFD to BSim will lead to errors in surface temperatures and energy flows throughout the building, causing errors in the boundary conditions supplied to CFD for the next time step. Clearly, an accurate treatment of internal surface convection is critical to the BSim-CFD integrated approach.

Treatment of Surface Convection in BSim-CFD

There are two basic options for modelling surface convection in the BSim-CFD simulator:

1) Have CFD calculate the air-to-surface heat transfer based on the CFD-predicted flow and temperature fields.
2) Have BSim calculate the heat transfer using empirical relations with surface-averaged and zone-averaged temperatures.

The first option is the more general and desirable approach: the calculations can respond to local flow patterns and local—rather than surface-averaged—heat transfer can be predicted.

But, the prediction of surface convection remains problematic for CFD, principally because of the nature of turbulence in room air flow and the related treatment of near-wall regions. The standard $k-\varepsilon$ turbulence model with log-law wall functions remains (by far) the most commonly employed approach in the CFD modelling of room air flow, although it has been well demonstrated that this can lead to significant errors in surface convection predictions (Chen and Jiang 1992; Yuan et al 1994; Awbi 1998).

Alternate turbulence models have been assessed and improved results have been observed, but only in some cases, and usually at the expense of higher compute requirements and/or stability (e.g. Chen 1995).

Research is underway to develop new methods to accurately resolve the wall heat transfer (e.g. Barp and Moser 1998, Xu et al 1998). Indeed, one new set of wall functions has been developed (Yuan et al 1994), although their applicability is limited to buoyancy-driven flow over vertical surfaces. These research efforts may one day result in robust and general methods to enable CFD to resolve wall heat transfer. However, until this time the second option outlined above remains the way forward: that is, for BSim to calculate surface convection using empirical relations with surface-averaged and zone-averaged temperatures.

Need for Improved Convection Modelling

The above discussion highlights the importance of improving the modelling of internal surface convection. The way forward, it is proposed, is to implement an "adaptive" algorithm to allow convection calculations to be responsive to local flow conditions. The BSim program would possess a suite of methods for calculating h_c, each one appropriate for a specific flow regime (e.g. buoyancy driven flow over walls, forced flow at ceilings). The program would select the appropriate approach for each internal surface, on a time step basis, based upon the configuration and the prevailing operational states. Such an approach is currently under development within ESP-r (Beausoleil-Morrison and Strachan 1999).

The ESP-r adaptive approach draws upon numerous h_c algorithms reported in the literature. However, an important flow regime—mixed convection in which both mechanical (fan) and buoyant forces are important—is not adequately addressed by existing h_c algorithms. Consequently a new method has been developed, the subject of the next section.

MIXED CONVECTION MODEL

Numerous algorithms exist for establishing h_c. Some are general in nature while the applicability of others is restricted to specific building geometries and plant systems. Most are simple in form, often regressions of empirical data which give h_c as a function of air and surface temperatures for a single flow regime.

The two methods that form the basis of the new mixed convection model are reviewed in this section. As space does not permit treatment of other approaches, the reader is referred to Beausoleil-Morrison and Strachan (1999) for descriptions of other h_c algorithms and further references.

Alamdari and Hammond Correlations

Alamdari and Hammond (1983) presented correlations for buoyancy-driven convective heat transfer for use in BSim programs. Correlations which cover laminar, transitional, and turbulent flow regimes for the following three configurations are given:

- Vertical surfaces.
- Stably-stratified horizontal surfaces (e.g. warm air above a cool floor).
- Buoyant flow from horizontal surfaces (e.g. cool air above a warm floor).

Rather than conducting new experiments, they drew upon data reported in the literature to develop their correlations, which are cast in a continuous form suitable for implementation into BSim programs. The relation for vertical surfaces, for example, is given by,

$$h_c = \left\{ \left[1.5 \cdot \left(\frac{\Delta T}{H} \right)^{1/4} \right]^6 + \left[1.23 \Delta T^{1/3} \right]^6 \right\}^{1/6} \qquad (2)$$

where ΔT is the air-surface temperature difference and H is the height of the vertical surface.

The correlations cover the full range of ΔT and dimensions relevant to building applications. However, there are limitations to their applicability:

- The correlations were generated from data derived from experiments on *isolated* or *free* surfaces, whereas air flow within rooms more closely approximates flow within an *enclosure*.
- The correlations are not applicable for mechanically driven jets as experienced in actively ventilated buildings, but rather are restricted to flow regimes dominated by buoyant forces.
- The correlations are only applicable for configurations in which buoyancy is a result of temperature differences between the room air and room surfaces. They are not applicable, for instance, for the flow regime generated by a warm plume rising from a radiator.

Fisher Correlations

Fisher (1995) performed experiments within a mechanically ventilated room-sized enclosure to develop correlations for internal surface convection. The experiments spanned a range of air flows and ventilation-air temperatures.

For the majority of the experiments the internal surfaces were held at the same temperature, the so-called *isothermal* room. In one group of experiments a single wall was chilled in order to examine the combined impact of buoyant forces against the wall and mechanical effects (the *non-isothermal* room). Convection correlations for three classes of flow were developed:

- Isothermal rooms with ceiling jets emanating from radial ceiling diffusers.
- Non-isothermal rooms with ceiling jets emanating from radial ceiling diffusers.
- Isothermal rooms with free horizontal jets emanating from wall air supplies.

The room's interior surfaces were covered by 53 panels, each an independent resistance-heater. Heat input to each panel was controlled to maintain the desired surface temperature. Surface convection was derived from these measurements by evaluating surface energy balances for each of the 53 panels. By maintaining all surfaces at the same temperature (in the isothermal cases), radiation exchange was minimized, thus reducing uncertainty in deriving the surface convection.

To minimize uncertainty the results were correlated with the ventilation-air temperature (T_{in}), rather than the room-air temperature (T_{air}) as in Equation 1:

$$h_c = \frac{q''_{conv}}{(T_{surf} - T_{in})} \qquad (3)$$

In the case of the isothermal ceiling jet—the most applicable configuration in the context of BSim—Fisher found, interestingly, that the surface convection was independent of the inlet velocity of the ceiling jet, but rather depended upon the jet's volumetric flow rate. He also found the buoyancy forces of the cold jet to be negligible. The form of the correlations[1], expressed in dimensionless parameters, reflect these observations. The relation for walls, for example, is given by,

$$Nu_{walls} = -24.8 + 0.36 \cdot \mathrm{Re}_e^{0.8} \qquad (4)$$

The Nusselt number is defined as $Nu = h_c V_{room}^{1/3}/k$ and the *enclosure* Reynolds number as $\mathrm{Re}_e = \dot{V}_{in}/\nu V_{room}^{1/3}$, where V_{room} is the room volume, k is the thermal conductivity of air, $\dot{V}_{in}$ is the volumetric air flow of the jet, and ν the kinematic viscosity of air.

These correlations were derived for the range of ventilation temperatures relevant for cooling ($10^oC \le T_{in} \le 25^oC$) and for a very large range of ventilation rates ($3 \le ac/h \le 100$; the data of Spitler et al 1991, acquired in the same experimental facility, were used for the higher flow rates).

Buoyant forces caused by temperature differences between internal surfaces and the room air were very small in the isothermal ceiling-jet experiments. Sixteen combinations of ventilation rate and temperature were assessed. In all cases, the internal surfaces were controlled to 30°C. The corresponding mean room-air temperatures (not reported) can be estimated by performing a heat balance on the room: the average surface-air ΔT over the 16 experiments was 2.4°C; the greatest difference was 3.8°C.

Although the non-isothermal experiments examined the combined impact of buoyant forces against the wall and mechanical effects, the range of temperatures examined was narrow and the combination of temperatures ($T_{coldwall} < T_{in} < T_{hotwalls}$) atypical.

Fisher's results represent a significant contribution to the modelling of internal convection in BSim. However, as with all approaches, there are limiting factors:

- The *isothermal* correlations are strictly applicable when the flow regime is dominated by a mechanically driven jet, buoyancy caused by surface-air temperature differences being negligible.
- The *non-isothermal* correlations are not generally applicable to mixed flow, wherein buoyant forces adjacent to some surfaces are important (e.g. a window exposed to the outside) and may assist or oppose the mechanical forces.

---

[1] Fisher and Pedersen (1997) present an alternate regression (in dimensional form) of the same experimental data.

- The room was cooled by the supply air in all experiments: forced-air heating systems were not examined[2].
- All experiments were carried out in a single room of constant dimensions, so the influence of room aspect ratio is unknown.

Mixed Flow: A Common Flow Regime

Two algorithms that BSim programs can use to calculate h_c on a time-step basis have been described. One (Alamdari and Hammond) is for purely buoyant flow (buoyancy caused by ΔT between room air and internal surfaces). The other (Fisher isothermal) is for purely mechanically driven jets. However, in mechanically ventilated rooms both forces will, in general, be present, and both can be significant. In some cases the mechanical and buoyant forces will assist (act in same direction) while in others they will oppose (act in opposite directions) or act transversely (act in perpendicular directions). Neither the Alamdari and Hammond nor the Fisher approach can fully characterize the convective regime in these mixed flow cases.

In addition, it is difficult (usually impossible) to predetermine whether a configuration will be dominated by buoyant forces or mechanical forces. This is best illustrated by example[3]. A well-insulated office with large glazing area is heated by a constant-volume forced-air system delivering 6 ac/h through ceiling-mounted diffusers. On a relatively cold day (-20°C) a supply air temperature of ~30°C is adequate to heat the office, the heating load being offset by solar gains and gains from lights, occupants, and office equipment. The internal surface of a wall facing the outdoors is about 4°C colder than the averaged room air temperature.

The warm jet emanating from the diffuser spreads across the ceiling towards the walls (it adheres to the ceiling rather than dropping due to viscous and buoyant forces). The jet cools as it flows down the outside-facing wall. Velocity is relatively low by this point due to the jet's spread. As the surface of the wall is colder than the surrounding room air, air adjacent to the wall contracts (becomes more dense) and sinks due to gravity: buoyancy assists the mechanically driven jet, both effects forcing flow down the wall.

Both mechanical and buoyant forces drive the air down the wall, but is the flow predominantly buoyant or mechanically driven? This question can be answered (qualitatively) by examining the surface convection predicted by the two approaches. Alamdari and Hammond (Equation 2) predicts surface convection to be ~$8W/m^2$; Fisher (Equation 4) predicts ~$7W/m^2$. As the predictions are of the same order, both buoyant and mechanical forces are considered significant. Since the forces are assisting, the surface convection should be higher than either method alone predicts. In this case, flow is mixed and the forces are assisting.

But during morning start-up (ie. recovery from night setback), the temperature of the supply air is much warmer (~40°C) while the wall-air ΔT is lower (~3°C). In this situation, Alamdari and Hammond gives ~$6W/m^2$ whereas Fisher gives ~$15W/m^2$. So in this situation the mechanical forces are dominant.

And on a sunny and relatively warm (-3°C) day the heating system supplies air just above the room air temperature and the wall-air ΔT is only ~2°C. In this situation, Alamdari and Hammond gives ~$3.5W/m^2$ whereas Fisher gives ~$1W/m^2$. In this situation buoyant forces are dominant.

The examination of other mechanically ventilated building configurations in different climates would, of course, lead to different observations. Although in general it would be seen that for some thermal and operational states, buoyant forces are dominant, while for others, mechanical forces are dominant, while yet for others both are important.

New Mixed Flow Correlation

A new model that blends the Alamdari and Hammond and Fisher algorithms is proposed for the general case of mixed flow. By considering the buoyant and forced correlations as asymptotic solutions, the following 3rd order sum gives h_c when the buoyant and mechanical forces are assisting or acting transversely (modelled after Churchill and Usagi's, 1972, general expression for correlating rates of heat transfer):

$$h_{c,mixed,assisting} = \left[(h_{c,Fisher})^3 + (h_{c,A\&H})^3 \right]^{1/3} \qquad (5)$$

When the buoyant and mechanical forces are opposing h_c is taken as the greater of the two predictions:

$$h_{c,mixed,opposed} = \max\left[h_{c,Fisher} \ , \ h_{c,A\&H} \right] \qquad (6)$$

This model reproduces identically the Alamdari and Hammond result when forced effects are unimportant and reproduces identically the Fisher result when buoyant effects are insignificant. When both effects are important and are assisting, Equation 5 results in a greater h_c than either method alone. Equation 6 ensures that h_c will not be lower than either method predicts.

---

[2] As buoyancy of the cold jet did not influence surface convection, a heating system using the same type of diffuser should in fact generate a substantially similar flow field. Therefore, it is felt that Fisher's correlations are equally applicable to room heating when there are negligible surface-air temperature differences.

[3] The temperature data cited in the example were acquired from an ESP-r simulation.

The mixed flow model has been incorporated into ESP-r, adding to the program's existing convection capabilities. Equation 5 is applied for all floors and ceilings, because on these surfaces buoyant forces always act in a transverse direction to the jet resulting from radial ceiling diffusers. For walls, a test is performed each time step to determine whether the wall-air ΔT results in a buoyant force that assists or opposes the mechanically driven jet, and correspondingly a decision made on whether to apply Equation 5 or 6.

A simulation with the mixed flow model was performed on the office previously described for the month of March using Ottawa weather data. Figure 1 plots q''_{mixed}, q''_{Fisher}, and $q''_{A\&H}$ for the outside facing wall. The surface convection heat flux is plotted against the surface-air ΔT to best illustrate the impact of the mixed flow model.

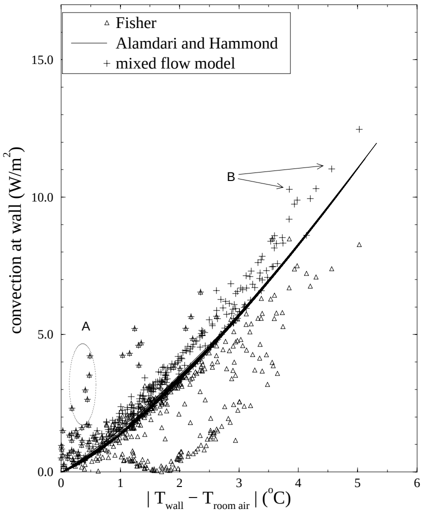

Figure 1: Constant Volume Heating

The Alamdari and Hammond model correlates well to ΔT, as expected (see Equation 2). However, the Fisher correlation does not, as it responds to T_{in}, a function of the room's heating load. At higher ΔT, Alamdari and Hammond tends to dominate, but at lower ΔT, where buoyant forces are small, q''_{mixed} approaches q''_{Fisher} (see region A in Figure 1). For a large number of data points q''_{mixed} is greater than both q''_{Fisher} and $q''_{A\&H}$, indicating that buoyant and forced effects are often both important (see B in Figure 1).

APPLICATION OF NEW MODEL

Description of Test Case

A two-zone (150 m^2 floor area per zone) ESP-r model representing one storey of a shallow floor-plate office building was created. The building, located in Ottawa, has a north-south alignment and is moderately glazed (35% of external wall area), all windows facing east or west. The fabric assemblies, insulation levels, and internal gains are typical of Canadian construction.

Each zone is conditioned with a constant-volume forced-air mechanical system whose supply-air temperature varies from 13°C to 43°C in response to loads. During occupied hours (5h00 to 20h00 weekdays) the system delivers 60 L/s of outdoor air to each zone. The building is heated to 22°C, with an 18°C setback during unoccupied hours. The cooling setpoint is 24°C while the building is allowed to free float during unoccupied periods in the summer. At 6 ac/h, the system is sized to meeting the peak heating load but is undersized for cooling.

Impact of Model on Load Predictions

Three annual simulations—identical except for the treatment of internal convection—were performed. The Alamdari and Hammond correlations were used in the first simulation (ESP-r's default approach), Fisher's correlations applied in the second, while the mixed flow model was utilized in the third[4]. The annual heating and cooling loads (normalized by floor area) are given in the following table.

h_c algorithm	annual loads heating (MJ/m^2)	annual loads cooling (MJ/m^2)
Alamdari & Hammond	271	207
Fisher	265	243
Mixed Flow Model	295	247

The mixed flow model predicts significantly higher heating loads than either Alamdari and Hammond (9% higher) or Fisher (11% higher). It also predicts substantially higher cooling loads than Alamdari and Hammond (19% higher) but only slightly more (<2%) than Fisher.

Impact on Thermal Comfort Predictions

Clearly, the choice of h_c algorithm has a significant impact on the prediction of annual heating and cooling loads (and thus energy consumption). Another (perhaps more) significant implication of algorithm choice can be seen by examining Figure 2, which plots the air temperature in the west zone on July 5, a day with very high cooling loads.

The system was not sized to meet the peak cooling loads, a valid design decision in a climate with a short cooling season; significant capital cost savings can be realized by sizing equipment to maintain the setpoint temperature through the majority of the cooling season, but allowing temperatures to rise on the most severe days. In such a case a designer might use BSim to assess whether thermal comfort will be unduly compromised by the undersizing.

[4] ESP-r's adaptive convection algorithm does not apply the Fisher or mixed flow equations when the forced-air system is inoperative; rather it switches to Alamdari and Hammond to more closely approximate the convective regime.

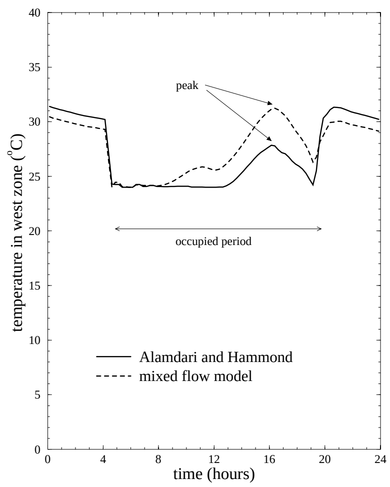

Figure 2: Mixed Flow Model

In this case, the ESP-r results indicate that although the setpoint temperature is maintained most of the time, there are a few problematic days. On July 5, for example, the system is unable to maintain the setpoint temperature, particularly in the afternoon when solar gains reach their peak. The temperature of the zone drifts upwards, reaching a maximum just after 16h00 (refer to Figure 2). When the Alamdari and Hammond correlations are used, a peak zone temperature of 27.9°C is predicted: this might be acceptable to the designer. However, when the mixed flow model is employed, a peak zone temperature of 31.2°C is predicted; this would be deemed unacceptable, leading the designer to alter the architectural and/or mechanical features of the building.

Impact on Assessment of Design Options

The designer might explore a number of options to address the overheating problem, including:

- Increasing cooling capacity of system by 50% by increasing flow rate from 6 to 9 ac/h.
- Increasing cooling capacity ~25% by lowering minimum supply air temperature from 13°C to 10°C.
- Changing system to VAV with a constant supply temperature of 13°C, a minimum flow of 6 ac/h, and a maximum flow of 9 ac/h, effectively increasing cooling capacity by 50%.
- Reducing solar gains by adding window overhangs.
- Pre-cooling the building by night purging with 6 ac/h with 100% outdoor air.

Each of these design options was simulated twice: first with the Alamdari and Hammond correlations and then with the mixed flow model. All measures reduced the peak zone temperatures, with varying degrees of success, and all had an influence on cooling loads, as shown in the following table (loads for month of July). The numbers in parentheses indicate the difference relative to the base design.

design option	**Alamdari & Hammond** cooling load (MJ/m^2)		**Mixed Model** cooling load (MJ/m^2)	
base design	51.8		58.5	
9 ac/h	52.5	(+1%)	61.6	(+5%)
10°C SAT	52.4	(+1%)	60.8	(+4%)
VAV	52.5	(+1%)	63.1	(+8%)
overhangs	41.2	(-20%)	48.7	(-17%)
night purge	42.8	(-17%)	51.3	(-12%)

The mixed flow model responds to changes in the flow regime (air change rates and supply air temperature); higher cooling loads are predicted for the first three design options, a result of the increased cooling capacity *and* increased surface convection. In contrast, the Alamdari and Hammond approach is not capable of responding to these changes in the flow regime. Consequently, cooling load predictions are only slightly higher (~1%) and due entirely to the fact that the cooling system was able to extract more energy because of its higher capacity.

In contrast, the Alamdari and Hammond correlations predicted greater savings with overhangs and night purging. These measures reduced cooling loads substantially with both convection methods, but the lower h_c produced in the Alamdari and Hammond runs overpredicted the savings.

CONCLUSIONS

More advanced and refined methods are required for modelling internal surface convection within BSim. The way forward, it is believed, is for BSim programs to adapt convection calculations to local flow conditions, an approach that requires BSim programs to be populated with h_c methods appropriate for various flow regimes. To this end, a new approach has been developed for calculating h_c for mixed convection within mechanically ventilated rooms, building upon Alamdari and Hammond's (1983) work on buoyancy-driven flow and Fisher's (1995) work on forced flow. The new method is applicable for rooms ventilated with ceiling mounted diffusers and is appropriate for both heating and cooling. It is suitable when the convective regime is dominated by a mechanically driven jet, when it is dominated by buoyant forces resulting from surface-air temperature differences, *and* when both effects are important.

In a series of ESP-r simulations of a mechanically ventilated office building, the mixed flow model predicted significantly higher heating loads (9% higher than Alamdari and Hammond; 11% higher than Fisher), indicating the importance of both buoyant and forced effects. Therefore, simulating configurations like this with a convection approach that considers only buoyant or only forced effects would significantly underpredict heating loads and heating energy consumption. The mixed flow model resulted in substantially higher cooling loads than Alamdari and Hammond (19% higher) but only slightly more (<2%) than Fisher, indicating that the convection regime is dominated by mechanical effects when the system is cooling (due to low surface-air ΔT). Therefore, cooling loads in cases like this could be accurately predicted with a convection approach that considers only forced effects, although an approach that considers only buoyancy effects would lead to significant errors.

Additionally, an improper choice of h_c algorithm could lead to inappropriate design decisions. Simulations of the office using the Alamdari and Hammond approach indicated that peak temperatures were borderline acceptable on the most severe cooling days; whereas, the mixed flow model clearly showed that comfort conditions could not be maintained. Simulations of a number of design measures aimed at mitigating the overheating problem illustrated that the mixed flow model can respond to changes in the flow regime resulting from mechanical system alterations, but the buoyancy-only approach cannot. The impacts of the five design measures assessed were found to be sensitive to the convection method utilized, demonstrating that the choice of h_c algorithm can influence design decisions drawn from a simulation-based analysis.

REFERENCES

Alamdari F. and Hammond G.P. (1983), ‘Improved Data Correlations for Buoyancy-Driven Convection in Rooms’, *Building Services Engineering Research and Technology*, 4 (3) 106-112.

Awbi H.B. (1998), ‘Calculation of Convective Heat Transfer Coefficients of Room Surfaces for Natural Convection’, *Energy and Buildings*, 28 219-227.

Barp S. and Moser A. (1998), ‘Heat Transfer Under Thermally Stratified Conditions at Horizontal Surfaces in Buildings’, *Proc. ROOMVENT ’98*, (2) 103-110, Stockholm Sweden.

Beausoleil-Morrison I. and Strachan P. (1999), ‘On the Significance of Modelling Internal Surface Convection in Dynamic Whole-Building Simulation Programs’, *accepted for ASHRAE Transactions*.

Chen Q. and Jiang Z. (1992), ‘Significant Questions in Predicting Room Air Motion’, *ASHRAE Transactions*, 98 (1) 929-939.

Chen Q (1995), ‘Comparison of Different $k-\varepsilon$ Models for Indoor Air Flow Computations’, *Numerical Heat Transfer*, B(28) 353-369.

Churchill S.W. and Usagi R. (1972), ‘A General Expression for the Correlation of Rates of Transfer and Other Phenomena’, *AIChE Journal*, 18 (6) 1121-1128.

Clarke J.A. (1991), *Internal Convective Heat Transfer Coefficients: A Sensitivity Study*, Report to ETSU, University of Strathclyde, Glasgow U.K.

Clarke J.A., Dempster W.M. and Negrão C. (1995), ‘The Implementation of a Computational Fluid Dynamics Algorithm within the ESP-r System’, *Proc. Building Simulation ’95*, IBPSA, 166-175.

Clarke J.A. and Beausoleil-Morrison I. (1997), ‘The Integration of Computational Fluid Dynamics into Building Simulation’, *IBPSA News*, 9 (2).

ESRU (1997), *The ESP-r System for Building Energy Simulations: User Guide Version 9 Series*, ESRU Manual U96/1, University of Strathclyde, Glasgow UK.

Fisher D.E. (1995), *An Experimental Investigation of Mixed Convection Heat Transfer in a Rectangular Enclosure*, PhD Thesis, University of Illinois, Urbana USA.

Fisher D.E. and Pedersen C.O. (1997), ‘Convective Heat Transfer in Building Energy and Thermal Load Calculations’, *ASHRAE Transactions*, 103 (2) 137-148.

Irving S.J. (1982), ‘Energy Program Validation: Conclusions of IEA Annex 1’, *Computer Aided Design*, 14 (1) 33-38.

Jones P.J. and Whittle G.E. (1992), ‘Computational Fluid Dynamics for Building Air Flow Prediction--Current Status and Capabilities’, *Building and Environment*, 27 (3) 321-338.

Judkoff R. and Neymark J. (1995), *International Energy Agency Building Energy Simulation Test (BESTEST) and Diagnostic Method*, IEA Energy Conservation in Buildings and Community Systems Programme Annex 21 Subtask C and IEA Solar Heating and Cooling Programme Task 12 Subtask B.

Holmes M.J., Lam J.K-W, Ruddick K.G., and Whittle G.E. (1990), ‘Computation of Conduction, Convection, and Radiation in the Perimeter Zone of an Office Space’, *Proc. ROOMVENT ’90*, Oslo Norway.

Spitler J.D., Pedersen C.O., and Fisher D.E. (1991), ‘Interior Convective Heat Transfer in Buildings with Large Ventilative Flow Rates’, *ASHRAE Transactions*, 97 505-515.

Waters J.R. (1980), ‘The Experimental Verification of a Computerised Thermal Model for Buildings’, *Building Services Engineering Research & Technology*, (1) 76-82.

Xu W., Chen Q., and Nieuwstadt F.T.M. (1998), ‘A New One-Equation Turbulence Model for Near-wall Natural Convection’, *Int. J. Heat Mass Transfer*, 41 3161-3176.

Yuan X., Moser A., and Suter P. (1994), ‘Wall Functions for Numerical Simulation of Turbulent Natural Convection’, *Proc. 10th International Heat Transfer Conf.*, Brighton UK, 191-196.