A MODEL OF A DISPLACEMENT VENTILATION SYSTEM SUITABLE FOR SYSTEM SIMULATION

Simon J.Rees and Philip Haves
Department of Building & Civil Engineering
Loughborough University of Technology,
Loughborough, LE11 3TU, UK

ABSTRACT

There is increasing interest in Europe in the use of displacement ventilation and chilled ceiling cooling systems. A modelling methodology is presented here that deals with the significantly different heat transfer characteristics of these systems compared with conventional all-air systems. The purpose of the work is to develop a room model that is computationally efficient enough for annual hourly simulation purposes and a nodal model has been developed that is intermediate in complexity between a single air node model and a CFD model. Results of steady state comparisons of the model predictions -and test chamber measurements are presented. A cooling system employing indirect evaporative cooling and displacement ventilation has been simulated using the model, and is included as an example of the simulation methodology. The model can also be extended to treat systems with chilled ceiling panels.

INTRODUCTION

Displacement ventilation systems have been in use in Northern Europe for more than a decade in office air conditioning systems and offer several potential benefits over conventional systems. These benefits include improved air quality in the occupant breathing zone, lower draught risk, and lower energy consumption. The heat transfer characteristics of these systems differ significantly from those of conventional systems with well mixed air distribution and require a more detailed model. The use of higher primary air temperatures in these systems, and the use of higher primary water temperature in chilled ceilings, suggest that some alternative central plant configurations may be advantageous, both in terms of lower energy consumption and avoidance of vapour compression refrigeration. The work described here constitutes the first stage of a project to develop suitable models of such systems for use in annual hourly simulations, to be used to analyse the performance of these systems with various central plant configurations in different climates and types of building.

In displacement ventilation systems for office spaces, air is supplied at low velocity at floor level at a temperature of 18–20°C. In systems without chilled ceiling panels, the supply air is heated as it flows across the floor — typically by 30–50 % of the difference between the supply and extract air temperatures [1] — so that the temperature of the air at ankle level is typically 21–22°C. Air and pollutant transport in the room is mainly by the thermal plumes developed over any internal loads. This generally upward movement of air and heat results in a warm layer of mixed air between the ceiling and the so called 'stationary front', as shown in Figure 1. The existence of these characteristic temperature gradients has implications for both thermal comfort and the heat transfer mechanisms in such rooms.

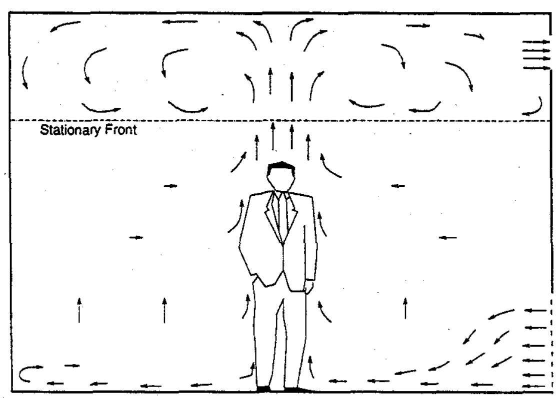

Figure 1: A simplified view of displacement ventilation flow

Thermal comfort considerations impose an upper limit to the allowable vertical temperature gradients in office spaces of 2–3 $K.m^{-1}$ [2]. Since these gradients are largely a function of internal load and room height, this results in a practical cooling load limit for these systems of about 30–40 $W.m^{-2}$. It is mainly for this reason that there has been growing interest in combining displacement ventilation with chilled ceiling panels in order to deal with loads up to about $80\,\mathrm{W.m^{-2}}$. These systems are also complementary in the sense that they can operate at similar primary water temperatures. The mean radiant temperature obtained with chilled ceiling panels is reduced compared to an all-air system [3, 4]), allowing the ventilation air to be introduced at a higher temperature ($\sim$ 1–2 K higher) for a given comfort level.

The heat transfer in rooms with displacement ventilation differs significantly from that in rooms with well mixed air. The warm air near the ceiling increases the surface temperature of the ceiling, which results in a net radiant heat transfer to the floor. This heat is then transferred by convection to the supply air as it flows across the floor. The resulting radiant asymmetry is an additional characteristic of rooms with displacement ventilation [5], but is not great enough to cause a comfort problem in itself [4]. Vertical temperature gradients along wall internal surfaces also occur, and under some external conditions it is possible for heat transfer through the walls to be inward at the bottom of the walls and outward at the top. Some measured air and surface temperature profiles from a test chamber the size of a two person office are shown in Figure 2. The effect of chilled ceiling panels is to reverse, or at least reduce, these vertical asymmetries.

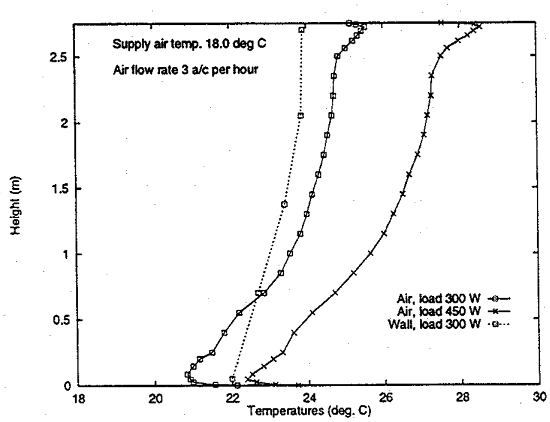

Figure 2: Room vertical temperature profiles measured by Li *et al*

MODELLING DISPLACEMENT VENTILATION SYSTEMS

Conventional building/system simulation programs typically assume that the room air is well mixed and can be defined by a single air temperature node. In order to model the principal heat transfer characteristics of systems incorporating displacement ventilation and chilled ceilings, it is necessary to increase the complexity of the room model. The issue then arises as to whether the air flow rates within the space should be specified as boundary conditions or whether they should be calculated by the model. The latter implies the use of CFD, which is too demanding computationally for use in annual hourly simulations, and so the approach adopted here is to use as much prior knowledge about the flow field as possible. As a first step, a thermal model with of the order of ten air nodes has been developed. The structure of the model reflects the known heat transfer paths and bulk air movements observed to occur in rooms with displacement ventilation. The air flow rates between the nodes have been determined by matching the predictions of the model with measurements made in a test room, as described below.

For pure displacement ventilation systems, it can be shown analytically [6] that the supply air mass flow rate required to maintain a constant height of the stationary front is proportional to the cooling load. If the temperature in the space is regulated by varying the supply flow rate (i.e. a VAV system), the height of the stationary front will remain constant, thus removing a degree of freedom that would otherwise require the model to have a much more flexible structure. This allows a model in which the air flow rate in each branch is a fixed fraction of the supply flow rate to represent a given room under varying load conditions, as long as the conductive heat transfer in the fabric is relatively small compared to the internal loads so that the external conditions have a small effect on the air flow within the room.

The model could be generalised by using measurements from different test rooms, together with results from CFD simulations, to derive empirical relationships that predict the air flow rates as a function of the geometry of the space, the ratio of the supply air mass flow rate to the cooling load, and the difference between the supply temperature and the ceiling temperature.

DEVELOPMENT OF A PROTOTYPE ROOM MODEL

A generalised nodal thermal modelling program, LIGHTS [7], has been used as a prototyping tool to develop a model of a two person office space with displacement ventilation. LIGHTS, which was originally developed to study the thermal interaction between artificial lighting systems and enclosed spaces, allows the user to specify a thermal network by defining nodes connected by conductances and/or capacity rates. Nodes may have thermal capacity and radiant exchange is treated using view factors. Boundary conditions are specified as known temperatures or heat fluxes at selected nodes. The program can produce either a dynamic or a steady state solution of the heat balance and (non-linear) rate equations.

The model is based on a test room equipped with a displacement ventilation system and a single heat source contained in a cuboidal enclosure [8]. The surface areas and view factors were defined from the geometrical description of the room. Several different configurations of nodes were implemented and the predicted air and surface temperatures compared with the corresponding measurements from the test room. The capacity rates and surface heat transfer coefficients were then varied in order to investigate their effect on the differences between the predictions of the model and the measurements. The aim was to explore the limitations of the different configurations rather than to determine the optimum set of parameters for a particular configuration.

The simplest nodal model tested consists of three air nodes representing the supply air, air near the floor and air near the ceiling (assumed to be the same as at the extract) and two surface nodes representing the ceiling and floor, as shown in Figure 3. The model was found to give reasonable predictions of the near floor and extract air temperatures but did not adequately define the temperature gradient in the room, and gave poor predictions of the wall surface temperatures.

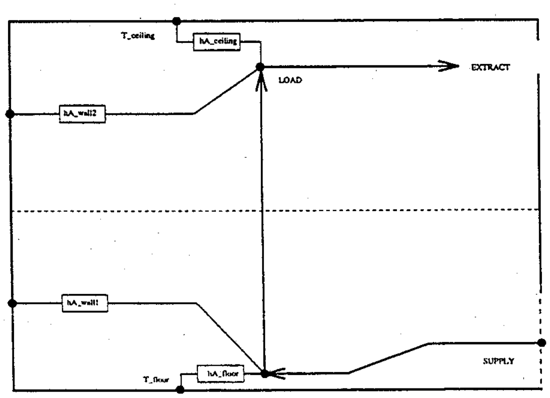

Figure 3: Simple nodal model of displacement ventilation

Nodal models with a single flow path represent a one-dimensional or 'plug flow'. However, in displacement ventilation systems, most of the heat from internal loads is transported directly to the upper zone of the room by buoyant plumes. The air flow rate and temperature in the plume depends on the rate of entrainment of air into the plume from the surrounding region. This concentration of the heat from the internal loads in the plumes can be thought of as introducing some decoupling of the walls from the loads. The room model has therefore been extended to treat the air flowing in the plumes separately from the surrounding air, as shown in Figure 4.

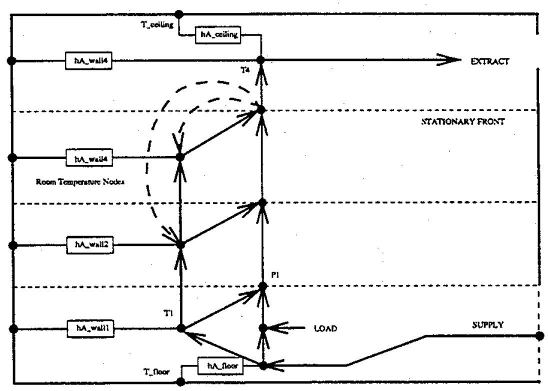

Figure 4: Nodal model with separate nodes for the plume

Two sets of air nodes have been introduced in the lower part of the room; one set representing the air flow upwards in the plumes, and a second set representing the air outside the plumes and in contact with the walls. The flow paths from the room air nodes to the plume air nodes represent the entrainment of air from the room into the plume. The walls are split into four vertically in order to better represent the air and surface temperature gradients. A further assumption in this model is that the upper most region in the room is well mixed, with the 'stationary front' three quarters of the way up the room.

Introducing these additional air nodes and flow paths also introduces extra parameters that have to be pre-determined, the most important being the initial split of the supply air between the load and the rest of the room. In principle this split, and the subsequent entrainment, could be calculated using a plume model for each category of internal load, although existing plume models do not treat the region below the top of sources of significant vertical extent, e.g. people.

One constraint on the division of the flow between the plume and the rest of the room arises from the second law of thermodynamics. The temperature in the plume cannot exceed the temperature of the heat source, which imposes a lower limit on the flow rate anywhere in the plume. In the experiments, the surface temperature of the enclosure containing the load was 50–60°C and it was found that a minimum of 20 % of the supply air flow had to be directed over the load in the model in order to obtain air temperatures at the load node consistent with the second law. The temperature profile predicted by the model in the lower part of the room was found to be sensitive to this parameter. In the absence of any experimental measurements of air temperatures near the load a value of 50 % was chosen for the percentage of flow directed over the load. The resulting room air and wall surface temperature profiles predicted by this model are shown in Figures 5 and 5 respectively.

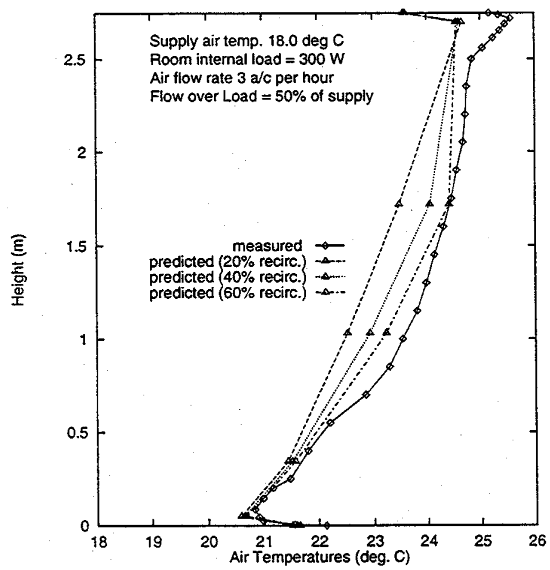

Figure 5: Comparison of air temperature predictions made by the model in Figure 4.

The more complex representation of the flow in this model, even with a crude estimate of the flow parameters, gives a more accurate estimate of the wall temperature profiles. In particular, the change in sign of the air-surface temperature difference between the top and the bottom of the wall seen in the experimental results is reproduced by the model. However, in order to obtain agreement between the predicted and observed air temperatures in the middle part of the room, it was necessary to introduce recirculation, as shown by the dashed lines in Figure 4. Some recirculation in the form of downward-flowing boundary layer currents along the walls is to be expected, since the walls are ~ 2 K cooler than the air in the upper half of the room. Further measurements are

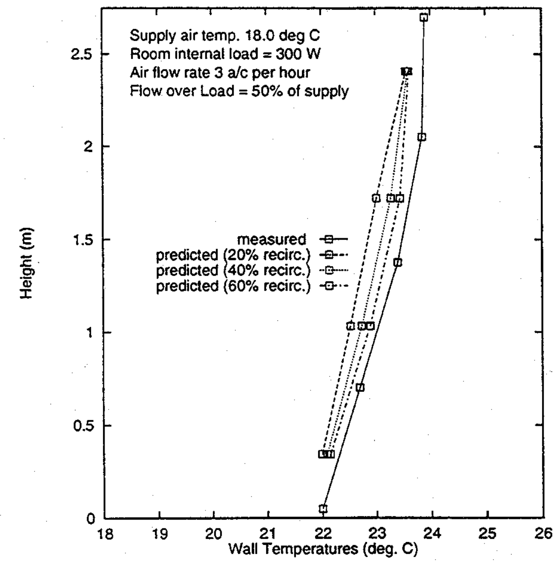

Figure 6: Comparison of wall temperature predictions made by the model in Figure 4.

required to determine whether the 60% recirculation required to achieve agreement between the model and the experiment actually occurs. An alternative interpretation is that the 'stationary front' is not well defined and that there is a region, starting approximately half way up the wall, in which the amount of recirculation progressively increases. The fact that the biggest temperature gradient occurs in the lower half of the room may indicate that the supply air flow rate is somewhat low for the size of the cooling load. The relatively low position of the load would also be expected to exacerbate this effect. The difference between the predicted and measured temperature profiles near the ceiling may be due to the strength of the plume from the concentrated load resulting in its direct impingement on the ceiling without significant lateral mixing.

The other principle parameters of the model that have to be determined are the surface convective heat transfer coefficients. Li *et al* [8] found that values in the range 5–7 $\mathrm{W.m^{-2}.K^{-1}}$ gave best results in their nodal models, whereas Mundt [9] found values in the range 3–5 $\mathrm{W.m^{-2}.K^{-1}}$ gave best agreement with experimental results. Initial tests of the models discussed here have been with values in the range 3–6 $\mathrm{W.m^{-2}.K^{-1}}$ and have shown that the results are not very sensitive to the values of the convective heat transfer coefficients. The simulations reported here have been carried out with constant coefficients of 6 $\mathrm{W.m^{-2}.K^{-1}}$ for floor and ceiling surfaces and 3 $\mathrm{W.m^{-2}.K^{-1}}$ for wall surfaces.

DEVELOPMENT OF A SYSTEM MODEL

A number of interesting system configurations involving displacement ventilation and chilled ceiling panels and their potential for energy saving are reported in the literature, both in the form of simulation results, and measurements from example projects [10, 11, 12]. It is planned to use the room model described here to study a variety of systems that have the potential to maximise free cooling and avoid the use of refrigeration plant. This section describes a prototype system model developed in the HVACSIM+ environment [13] using a version of the LIGHTS program that has been implemented as a HVACSIM+ component model [14]. The configuration used in the simulation is illustrated in Figure 7.

The supply air is cooled by indirect evaporative cooling, as shown in Figure 7. The return air is humidified adiabatically and then cools the supply air via a cross-flow plate heat exchanger. The temperature of the supply air is regulated by varying the effectiveness of the humidifier using a PI controller. The supply air flow rate used in the prototype simulation corresponds to three air changes per hour and the internal load is 300 W ($\sim 20\,\mathrm{W.m^{-2}}$), as in the steady state model testing. Since the load in the room remains essentially constant in this simple case, no feedback control of room temperature has been implemented. This simple plant arrangement and control strategy was chosen as a starting point for studying summer cooling performance.

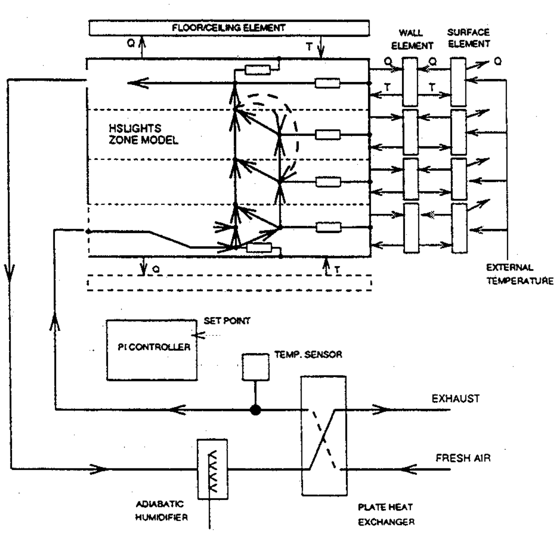

Figure 7: Configuration of the prototype system and room model

The HVACSIM+ version of LIGHTS is slightly different to the stand-alone version, in that only the internal surfaces of the zone fabric are included in the model, allowing conduction and thermal storage in the fabric to by treated by other component models in the simulation. This allows a flexible approach, so that a number of instances of the zone model can be linked to different configurations of fabric elements within a simulation. A single instance of a simple mass wall model has been used to represent a common floor/ceiling. The upper side is coupled to the floor surface and the lower side is coupled to the ceiling surface in order to model the effect of similar zones above and below. Heat transfer through one of the longer zone walls is modelled by four mass walls and external surface model units, the other walls being treated as adiabatic surfaces.

The heat exchanger is modelled using a fixed effectiveness (0.85) and its dynamic response is ignored. The humidifier is modelled using a variable effectiveness, defined as the ratio of the (dry bulb) temperature drop across the humidifier to the difference between the entering dry bulb and wet bulb temperatures. The sizing of the humidifier is treated by limiting its maximum effectiveness to 0.9. Both models ignore dynamic effects. The only dynamic model, apart from the PI controller, is the temperature sensor model, which is linear and first order, with a time constant of 60 seconds.

Since a first order system is stable under proportional control with an arbitrarily large proportional gain, it is not strictly necessary to use integral action to remove offset errors. However, use of a large proportional gain increases the stiffness of the resulting equation set, with a consequent increase in the computing effort required. The tuning values used were derived using the Ziegler-Nichols open loop method [15] by assuming that the effective time delay is equal to the time constant of the temperature sensor, effectively making some allowance for the neglected dynamics of the heat exchanger and the humidifier. It should be noted that the tuning values are not critical since the disturbances to be rejected by the controller are determined by hourly weather data and hence are of low frequency.

PROTOTYPE SIMULATION RESULTS

Results are presented here for a single day selected as being reasonably representative of a moderate summer day for the UK. The data were measured at Kew (near London) on August 7th, 1967.

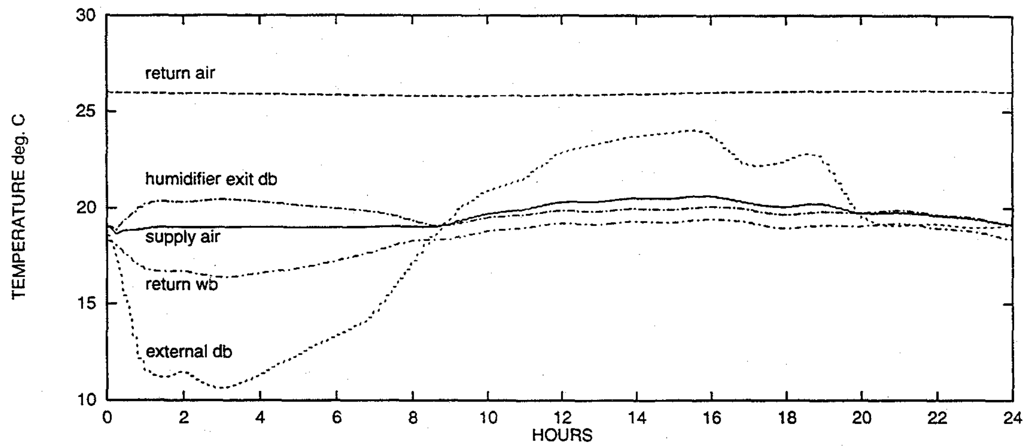

Figure 8: Simulation results for typical UK summer day

The dry bulb temperature is in the range 10.6–23.9°C, and humidity ratio is in the range 0.0083–0.0114 kg.kg^{-1}. The set-point for the supply air temperature was maintained at 19°C throughout the test period. The results of the simulation over the twenty four hour period are given in Figure 8.

The supply air temperature can be seen to follow the set-point during the early hours of the simulation but rises to nearly 21°C at about 16.00 hrs — the depression below the outside dry bulb being ~ 2 K. From 8.00 hrs the effectiveness of the humidifier is at its maximum and the cooling of the supply air is limited by the return wet bulb temperature — the supply air temperature profile follows that of the return wet bulb temperature with an offset of 1.5–2 K. Variations in the zone return air temperature are strongly damped by the thermal mass of the zone model under the constant load conditions simulated in this case.

During the first eight hours of the simulation, the humidifier operates even though the outside dry bulb temperature is below the set-point for the supply air temperature. This could be avoided by the use of face and by-pass dampers on the heat exchanger to limit the heat transfer from the return air to the supply air.

CONCLUSIONS

A model of the thermal behaviour of displacement ventilation systems has been developed based on a single set of experimental measurements and incorporated in a component-based simulation program. The behaviour of a room with displacement ventilation supplied by an indirect evaporative cooler has been simulated to illustrate the use of the model to assess system performance.

Further work will be concentrated in three areas:

- extension and validation of the room model
- more detailed system simulation
- implementing the room and plant models in a more computationally efficient simulation program

The extension and validation of the model will be performed using coupled CFD and radiant exchange programs [16, 17] and measurements from other test rooms. The aim is to produce a model that can treat the behaviour of properly designed and operated displacement ventilation systems, with and without chilled ceilings, at varying loads. The effect of walls in inducing recirculation will be investigated, with the intention of producing a model that can treat narrower spaces, such as cellular offices, as well wider spaces, such as open plan offices.

More detailed system simulation will involve realistic scheduling of heat gains, including solar gains, modelling of cooling towers as a source of water for chilled ceiling panels, and modelling of water-coupled floor/ceiling slabs to provide diurnal thermal storage.

LIGHTS and HVACSIM+ constitute a powerful prototyping environment but the speed of execution is much slower than hourly building energy analysis programs such as BLAST and DOE-2. The possibility of incorporating the displacement ventilation model in IBLAST (a version of

BLAST which performs a simultaneous solution of the building loads and HVAC system [18]) is currently being investigated.

ACKNOWLEDGEMENTS

The authors would like to thank Y. Li for provision of experimental data and helpful discussions during the model development, E. F. Sowell for his support in the use of the LIGHTS and HSLIGHTS programs, and J. J. McGuirk for his comments on the modelling methodology. This project is jointly funded by British Gas plc, EA Technology, the Ove Arup Partnership and Loughborough University of Technology.

REFERENCES

[1] P.J. Jackman. Displacement ventilation. In *Proc. of CIBSE national conference, Canterbury*, pages 364–380, 1991.

[2] ISO Standard 7730. *Moderate thermal environents—determination of the PMV and PPD indices and specification of the conditions for thermal comfort.* International Standards Organisation, Geneva.

[3] M. Udagawa. Simulation of panel cooling systems with laminar subsystem model. *ASHRAE transactions*, 99(2):534–547, 1993.

[4] R.W. Külpmann. Thermal comfort and air quality in rooms with cooled ceilings — results of scientific investigations. *ASHRAE Transactions*, 99(2):488–502, 1993.

[5] Y Li, L Fuchs, and M Sandberg. Effects of thermal radiation of airflow with displacement ventilation: an experimental investigation. *Energy and Buildings*, 19:263–274, 1993.

[6] P.F. Linden, G.F. Lane-Serff, and D.A. Smeed. Emptying filling boxes: the fluid mechanics of natural ventilation. *Journal of Fluid Mechanics*, 212:309–335, 1990.

[7] E.F. Sowell. *LIGHTS User's Guide.* Department of Computer Science, California State University. CA, USA., 1989.

[8] Y Li, M Sandberg, and L Fuchs. Vertical temperature profiles in rooms ventilated by displacement: Full-scale measurement and nodal modelling. *Indoor Air*, (2):225–243, 1993.

[9] E Mundt. Convection flows above common heat sources in rooms with displacement ventilation. In *Proc. of 'ROOMVENT-90', Oslo, Norway.*, 1990.

[10] W. Braun. Free cooling with displacement ventilation. In *Proc. of 'ROOMVENT-90', Oslo, Norway.*, 1990.

[11] M.F. Brunk. Cooling ceilings — an opportunity to reduce energy costs by way of radiant cooling. *ASHRAE Transactions*, 99(2), 1993.

[12] U. Busweiler. Air conditioning with a combination of radiant cooling, displacement ventilation, and desicant cooling. *ASHRAE Transactions*, 99(2):503–510, 1993.

[13] D.R. Clark and W.B. May. *HVACSIM+ Building System and Equipment Simulation Program - Users Guide.* National Bureau of Standards, 1985.

[14] E.F. Sowell. *A general zone model for HVACSIM+: Users manual.* Department of Engineering Science, report No. 1889/91, University of Oxford., 1991.

[15] J G Ziegler and N B Nichols. Optimum settings for automatic controllers. *Transactions of the ASME*, 64:759–768, 1942.

[16] CFDS. *CFDS-FLOW3D Release 3.3 User Guide.* AEA Technology, Computational Fluid Dynamics Services., Harwell, Didcot, Oxfordshire., 1981.

[17] CFDS. *RAD3D Release 1.4 User Guide.* AEA Technology, Computational Fluid Dynamics Services., Harwell, Didcot, Oxfordshire., 1981.

[18] R.D. Taylor, C.O. Pedersen, and L. Lawrie. Simultaneous simulation of buildings and mechanical systems in heat balance based energy analysis programs. In *Proc. of 3rd Int. Conference on System Simulation in Buildings, Liege, Belgium*, 1990.