VALIDATION OF A COMPUTATIONAL FLUID DYNAMICS SIMULATION OF A SUPPLY AIR ‘VENTILATED’ WINDOW

Dr M. McEvoy[1] and R.G. Southall[1]

[1] Department of Architecture, Cambridge University, UK.

ABSTRACT

A currently unresolved problem in building design is the paradox between increasing demand for good thermal insulation, and the requirement for ample levels of ventilation, to maintain a healthy indoor environment. A possible solution to this problem is a supply air ‘ventilated’ window. This utilises an airflow between panes to pre-heat ventilation air to the building, and to reduce thermal convection losses thus reducing the window U-Value. At the base of the window is a vent to the external environment, allowing air inflow. This relatively cold air is heated by convection/conduction from the warmer inner pane and will subsequently rise, or be drawn up, entering the room through venting at the top of the glazing.

A 2 Dimensional Computational Fluid Dynamics (CFD) model, using a commercial package optimsed for modelling the built environment, has been constructed to model the performance of the window. This model is described and will be validated against ASHRAE standard U-Values for conventional windows, and against experimental measurements taken from a Supply Air Window test rig that we have constructed. The simulation achieves a good degree of accuracy for steady state conditions over a wide range of varying parameters; ventilation rate, indoor outdoor temperature differentials, glass type, and cavity, and proves itself to be a useful tool in the design of Supply Air ‘Ventilated’ windows.

KEYWORDS

Ventilation, Energy Conservation, Window, Mass Transfer, Air Transport

VALIDATION OF A COMPUTATIONAL FLUID DYNAMICS SIMULATION OF A SUPPLY AIR ‘VENTILATED’ WINDOW

ABSTRACT

A currently unresolved problem in building design is the paradox between increasing demand for good thermal insulation, and the requirement for ample levels of ventilation, to maintain a healthy indoor environment. A possible solution to this problem is a supply air ‘ventilated’ window. This utilises an airflow between panes to pre-heat ventilation air to the building, and to reduce thermal convection losses thus reducing the window U-Value. At the base of the window is a vent to the external environment, allowing air inflow. This relatively cold air is heated by convection/conduction from the warmer inner pane and will subsequently rise, entering the room through venting at the top of the glazing.

A 2D Computational Fluid Dynamics model, using a commercial package optimsed for modelling the built environment, has been constructed to model the performance of the window. This model is described and will be validated against literature quoted standard U-Values for conventional windows, and against experimental measurements taken from Supply Air Window test rigs that we have constructed. The simulation achieves a good degree of accuracy for steady state conditions over a wide range of varying parameters including window size, ventilation rate, indoor outdoor temperature differentials, glass type, and cavity, and proves itself to be a useful tool in the design of Supply Air ‘Ventilated’ windows.

INTRODUCTION.

The need to reduce energy consumption has, in the last few years, become clear. With 42% [1] of the UK’s end use energy going into the heating of domestic and commercial properties, improved building thermal insulation should be a priority.

A Supply Air ‘Ventilated’ window has been shown to be effective in reducing window U Value [2], [3] and [4], and pre-heating the ventilation airflow. In new super-insulated houses that have catered for the required ventilation levels, air ingress is now the single largest source of building heating demand [5]. The window consists of a multiple glazed window with airflow between two of the panes. Air enters the cavity from a vent to the outside at the bottom and is drawn into the room at the top. See Figure 1. When the outdoor environment is significantly cooler than the indoor, heat convected between the panes of glass will be picked up by this column of air and transported into the room reducing the window U-Value. At times of high incident solar irradiation this method can be used to deliver warm air flow to the building instead of high localised radiative heat fluxes, which can lead to thermal discomfort for the occupants.

However, previous work has not investigated all the factors that determine the window performance and a comprehensive simulation is therefore desirable. Also it would appear that CFD has never been used in the simulation of this type of window, even though it would appear to lend itself well to describing the processes involved. This work is the first step to building a comprehensive model that will allow analysis of all aspects of the window performance, and the factors that affect it.

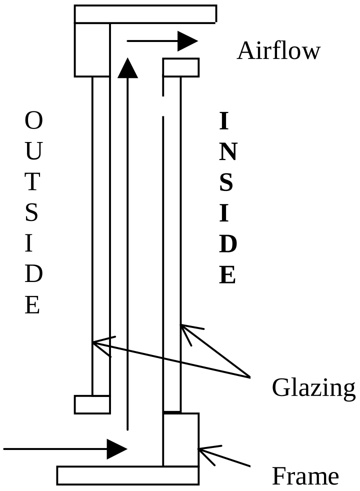

Figure 1. Principle of Supply Air Window Design

The standard measure of a window's thermal insulation properties is the U Value, in Watts/m^2.K. However this is conventionally the heat from the building entering the window. This is however not valid for a Supply Air 'Ventilated' window as it ignores the heat reclaimed in the ventilated cavity. We therefore use the term Effective U Value (Ue) which denotes the heat lost to the exterior from the outer pane and is the same as the U Value for an unventilated window.

EXPERIMENTAL METHOD

An experimental rig was designed and built to study the effect of cavity width and temperature differentials on the performance of a supply air ventilated window. The design of the window is shown in figure 2, and a photo of the experimental set-up is shown in figure 3. The knobs that can be seen at the 4 corners of the window in figure 2 can be turned to vary the distance from the stationary pane. This allows the cavity width to be adjusted from 0 to 50 mm, which was seen to be a practical range of cavity width, (12 to 14mm is a standard cavity width in double or triple glazed windows). We did not go below 10mm as a powerful ventilation system is required to drive reasonable ventilation rates though the window at such low cavity widths. From equations in [6] we have calculated that if a Passive Stack System was to be used, a 17m high one would be required to ventilate at 5mm gap width, compared to 3m for a 10 mm gap.

Measurement of the cavity width is done via observation through glass panels set into the side of the apparatus. The size of the window is 1.05 m by 1.05 m. Above and below the window is a hollow box that allows the air to settle slightly before leaving through a circular spigot 200 mm in diameter. Air comes in through a similar spigot at the bottom of the window. The spigots were originally intended for the attachment of tubing between the inlet and outlet, with a fan in-between. This forms a closed circuit through which smoke could be pumped to determine the turbulent state of the airflow.

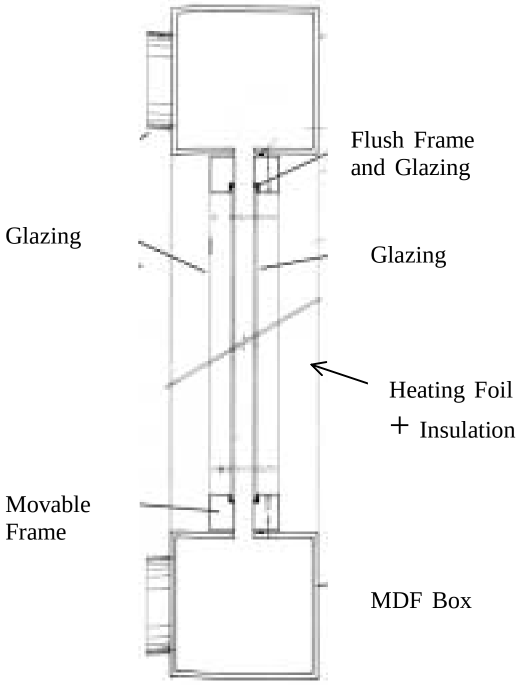

Figure 2. Experiment Schematic

Figure 3. Photo

To simulate the operating conditions of a supply air window, one pane had to be heated to form a temperature differential between the panes. This corresponds to an indoor/outdoor temperature differential that would exist between a heated room and a relatively cold external environment. This was achieved by using a sheet of heating foil taped to the stationary pane, and then backed by a sheet of MDF. Mains voltage was applied across the foil which then delivers 350 W of heating at 240V. The voltage delivered to the foil was varied by a variac to give different levels of heating.

Thermocouples were placed in the air inlet, and at 5 positions on the outside moveable pane, at distances of 7, 14, 21, 42, 63, and 84 mm from the bottom of the window to measure how the air in the cavity heats as it rises. The thermocouples were concentrated at the bottom of the window for better measurement of the initial and rapid temperature increases at the base of the window. Three thermocouples were placed between the steel and the heating foil to measure the heated pane temperature. No thermocouples were placed within the cavity to avoid disturbance of the airflow within the window. Two thin film low flow anemometers were placed at the top of the cavity, pointing down to measure airflow velocity, and as they are temperature corrected, the temperature as well. The anemometers were placed at a distance of 10 cm either side of the centre of the window. The position of these anemometers was changed to measure velocity profiles across the cavity.

Heated pane temperatures of 66 ^{o}C and 55 ^{o}C were achieved. This lead to temperature differentials of 44^{o}C and 33^{o}C respectively. These are higher differentials than would normally be encountered, but the higher the differential the more significant is the heating of the airflow, and the effects become easier to measure. All parameters were corrected (e.g. density, viscosity) for these higher temperatures so the results are valid for more normal conditions. This experiment in no way tries to mimic real life situations, but is used to study the effect of cavity width and turbulence levels on U-Value in a comparative manner.

SIMULATION METHOD

CFD is a method of solving the physical laws of conservation for heat and mass transfer. The conservation laws of mass, momentum, and energy must be solved for a situation such as ours which involves both fluid flow and heat transfer. All CFD solutions are based on the same basic principles [7].

- Approximation of the unknown flow variables by means of simple functions
- Discretisation by substitution of the approximation into the governing flow equation, and subsequent mathematical manipulations.
- Solution of the algebraic equations.

Various methods, Spectral, Finite Difference, Finite Element and Finite Volume are used to complete the first two steps. The Finite Volume method is the most common and well-validated technique, which forms the basis for most commercially available CFD codes. Our CFD software employs the finite volume method for solution of the governing equations. The Finite Volume method is distinct from the others as the integration of the governing equations takes place over finite control volumes within the solution domain. This process expresses the conservation of relevant properties for each finite cell size. It is easy to relate the concept of the conservation laws with a finite volume or box, and this is one of the main reasons why Finite Volume methods have proved so popular as the underlying concepts are easier to understand than Spectral or Finite Element methods.

The turbulence model employed by our software is the κ-ε model. Two extra differential equations now need to be solved, for the turbulence kinetic energy (κ) and it's rate of dissipation (ε). This is again the most widely validated method, and shows good performance for most practical applications. However poor performance is reported in non-circular ducts [7], which is the case in the window cavity.

First, simple models of normal unventilated windows were built. This was done as standard U-values are available from published sources for comparison. Also, many of the heat transfer processes that occur in unventilated windows occur in ventilated ones as well, e.g. radiative exchange with the environment, convective/conductive heat transfer from the outer panes to the internal/external air, glass conduction etc. It is therefore useful to know that the CFD methods we are using model these processes well.

We are primarily interested in centre of pane glazing, as inclusion of the frame introduces too many variables for effective comparison. We can therefore model the window in 2 dimensions, as this reduces computation time, and does not effect the results in the steady state. The glass is modelled as normal clear float glass with an emissivity of 0.84, with cavities of 12mm (most common gap width), and low E coatings of 0.17 emissivity.

To simulate the indoor and outdoor environment, the solution domain is split into two. Boundary walls with high U-Value then surround the outdoor domain, and a constant outdoor temperature set on the outer sides of these boundary walls. The internal environment is also surrounded by boundary walls with a U-Value representative of building fabrics of 0.6, and an internal temperature of 20 °C. The U-Value of the wall between the two domains is set to 0. The thermal conductivities, densities, specific heat capacities and emissivities of the glass types used in a typical window construction are taken from [7]. Inter-pane, internal and external radiative exchange is modelled. In the case of the latter two by panels placed on each end of the domain, facing the window one at external temperature and one at the internal temperature. Solar irradiation cannot however be modelled in conducting surfaces with this software version. U-Values were determined by summing the heat flow passing out of the external environment domain which comes solely from heat passing through the window.

We then created a model of the test rig set-up at the University of Westminster. From the previous description of the experiment it is clear that there is now no need for two separate domains. The rig is now modelled within one large domain (the Room). We have attempted to recreate the features of the experiment as much as possible. The film (with a heating power of 350W) has been simulated, as well as the metal sheet, insulating MDF panel/structure, and the glass itself. All with standard coefficients of thermal conductivity, density and specific heat capacity. The glass has also been modelled for radiative heat exchange, between the panes and with the external room, with the use of standard glass surface emissivities of 0.84. Below are pictures of the experimental structure, with temperature bands shown and a close up with air velocity vectors for the free convective case, at a cavity width of 50mm. Figures 15 and 16.

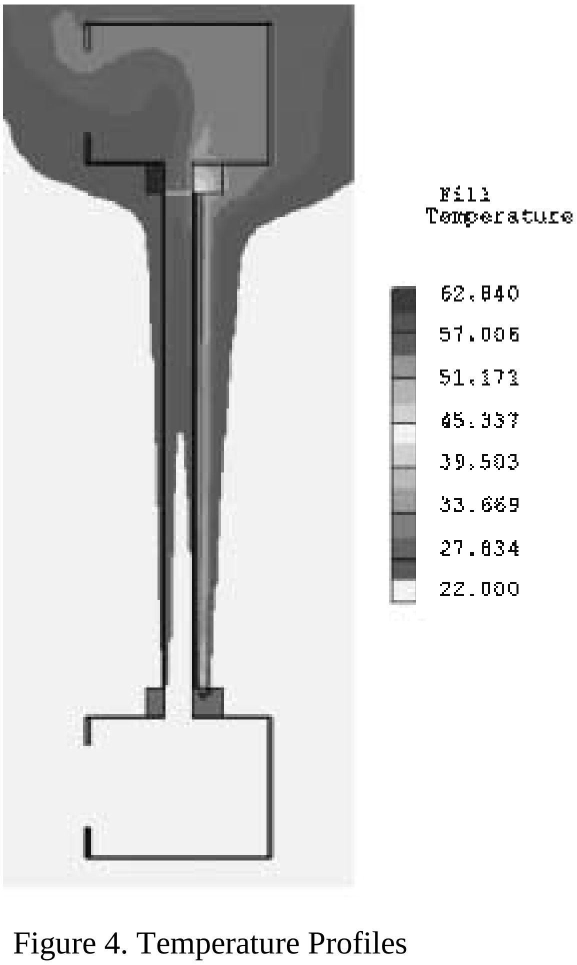

Figure 4. Temperature Profiles

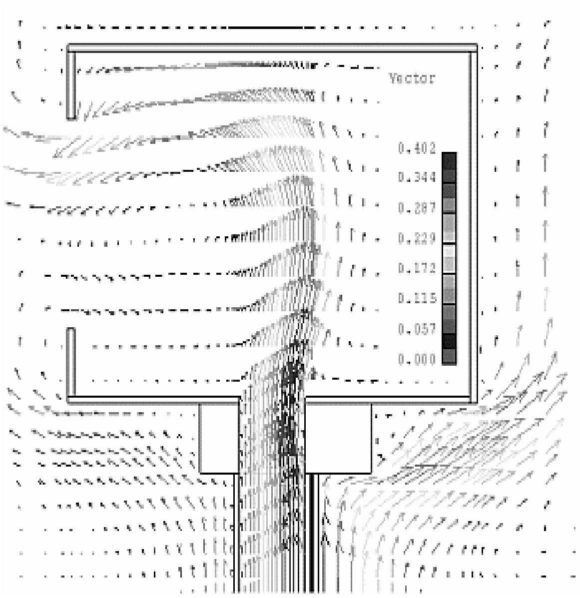

Figure 5. Velocity Vectors,

Within the simulation, monitored points are set up at the corresponding locations to the measurement devices in the experiment.

RESULTS

Shown below in Figure 6 is comparison of quoted U-Values, [8] [9], and the simulation for single, double and triple glazed windows with or without low E coatings.

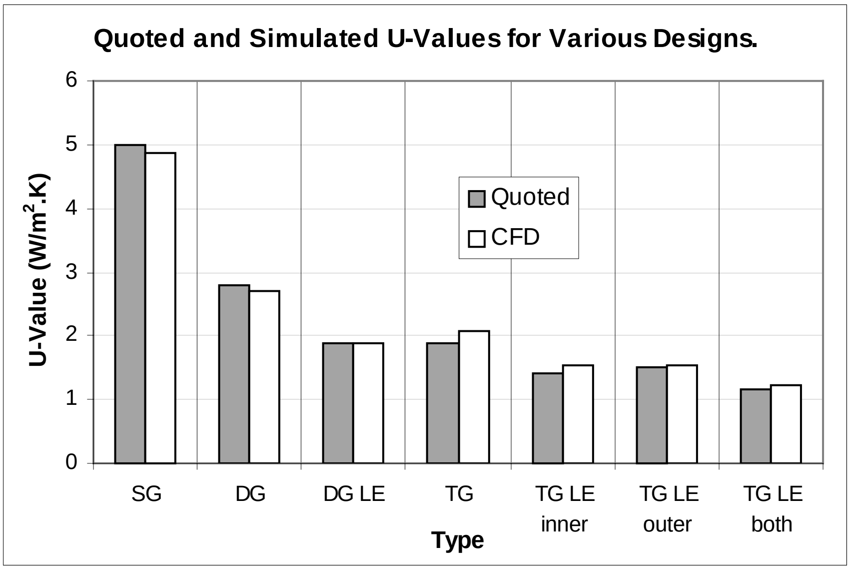

Figure 6. Unventilated Window U-Values

Agreement is in general excellent, especially as data on window U-Values is normally only given to the nearest 0.1 There does seem to be a slight tendency to underestimate high U-Values, and over estimate low u-Values, but the effect is small. The largest error is for the simple triple glazed window case with a 0.16 differential, or 8.4% of the whole heat transfer value.

Comparison of data from the simulation and the experiment are shown below in Figures 7 & 8. In Figure 7 the temperature readings from the low velocity anemometer (LVA), situated in the centre of the cavity at the top of the window, are compared with simulation, and in 8 the velocity readings from the LVA are compared for the free convective case.

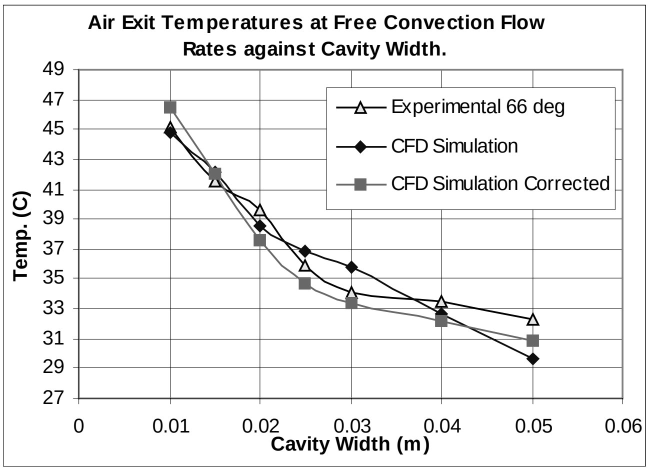

Figure 7.Air Exit Temperatures.

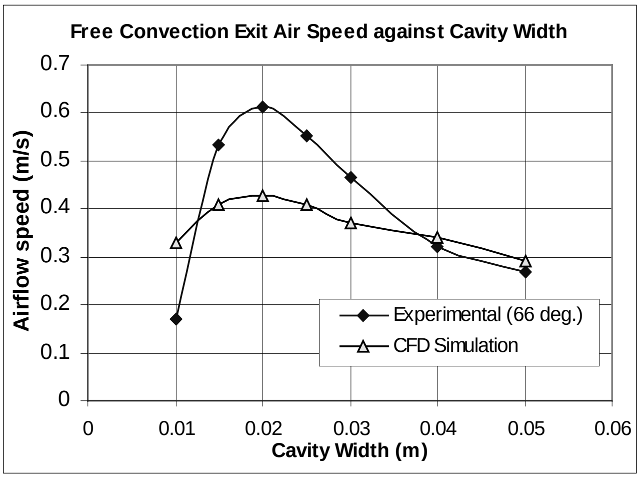

Figure 8. Air Flow Velocities

Air exit temperatures compare fairly well, but the simulation does not appear to be able to recreate the appropriate levels of free convective flow within the cavity. This is possibly due to the fact that the CFD code we are using is optimised for the large spaces often found in the built environment, and the equations governing airflows near to surfaces are optimised for this, and not the narrow gaps we are modelling here. The standard κ–ε model that we are using is not suited to non-circular ducts, and it is unclear if wall functions to model flow near walls more accurately have been included. When correcting for this, and simulating the correct fluid speed, we get the improved temperature profile shown in graph 1.

Outer pane temperature, which determines the Ue for the window, is an important parameter that must be simulated well. An example is shown below in Figure 9 of the 25mm cavity width, at 3 airflow speeds, 0.288 m/s, free convection at 0.55m/s and 1.04 m/s.

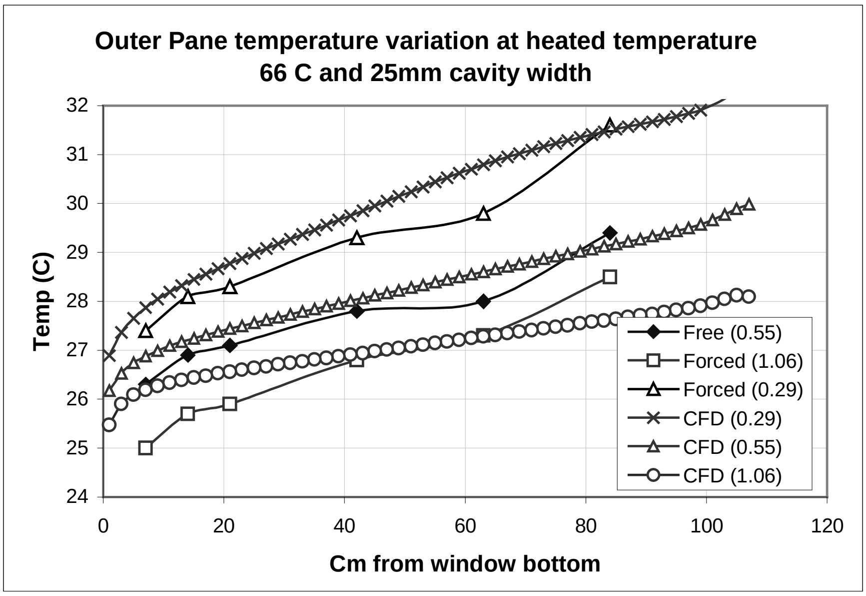

Figure 9. Outer Pane Temperatures

Agreement is very good, with almost all simulated points within 1 K of the experimental. We would therefore expect to get good agreement on the window Ue Value. This was proved to be the case, as can be seen in Figure 10 (again the 66°C case), with a heat transfer coefficient of 8 w/m^{2}K to calculate the experimental values.

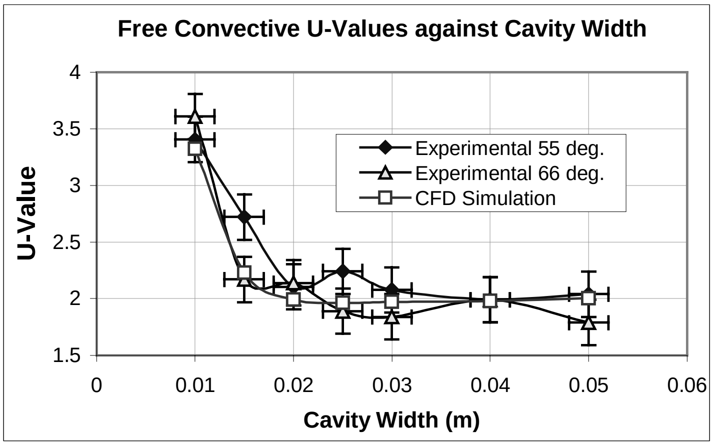

Figure 10. Ue Values

Agreement is very good, and with the variability in the experimental values, there is obviously significant experimental error present. However the level of U-Value, and the trend with cavity width is well modelled. Similar results are seen for all the cavity widths and ventilation rates. It should be noted however that these Ue-Values are higher than we would actually expect in a real window situation. The direct heating of the ‘inner’ pane allows no downdraught of colder air on the inner side, as would normally be the case, which acts to insulate the window further. We would therefore expect to see lower U-Values with a real ventilated window. This has proved to be the case with the other experiments of actual ventilated windows that we are doing.

It would appear that CFD simulates adequately the thermal processes within the window under forced convection conditions (which would be typical in real applications), however significant disparity is present when modelling free convective flows. Further study is therefore planned into the ability of different turbulence models and wall functions to model this situation adequately.

CONCLUSIONS

CFD simulates the general window concept very well, and the ventilated case to a good degree of agreement. The only source of significant error is with the free convective flow rates within the cavity, although this may not be the case with different turbulence models, or $\kappa-\varepsilon$ with wall functions included. We will be examining this with the next version of the CFD software. When this effect is corrected for the agreement with the experimental data is better, and there seems to be no problem modelling forced convective flows with CFD. We are now in a position to use these techniques to examine in detail the processes going on in the window. Further validations will take place with experimental data from test cells where we have installed various configurations of ventilate d window. One will analyse the effects cavity width and glass type, the other will analyse the effect of window size and solar irradiation. With this data and the subsequent model validation, we will have a picture of how all the relevant factors contribute to the window performance, and will allow from a thorough design optimisation.

ACKNOWLEDGEMENTS

We would like to thank the EPSRC for funding this research.

REFERENCES

[1] Boyle, G. (1996). Renewable Energy. Oxford University Press.
[2] Yuill, G K. (1987). Laminar Airflow Super Window. Renewable Energy Branch, Energy Mines and Resources. Canada
[3] Barakat, S A. (1987). Thermal Performance of a Supply Air Window. Proceedings of the 12th Annual Passive Solar Conference, Volume 12, pp 152-158.
[4] Tjelflaat,P O, Bergesen, B. (1985). Improved Thermal Insulation in Windows by Laminar Airflows. Thermal Performance of the Exterior Envelopes of Buildings III, pp 992-1003

[5] Roaf, S and Hancock, M. (1992). Energy Efficient Building. Blackwell.
[6] Awbi, H. B, (1991). Ventilation of Buildings. E & FN SPON.
[7] Versteeg, H. K, Malalasekera, W. (1995). An Introduction to Computational Fluid Dynamics. Longman Scientific and Technical.
[8] ASHRAE Fundamentals. 1995
[9] Pilkington Glass UK. Technical brochure.