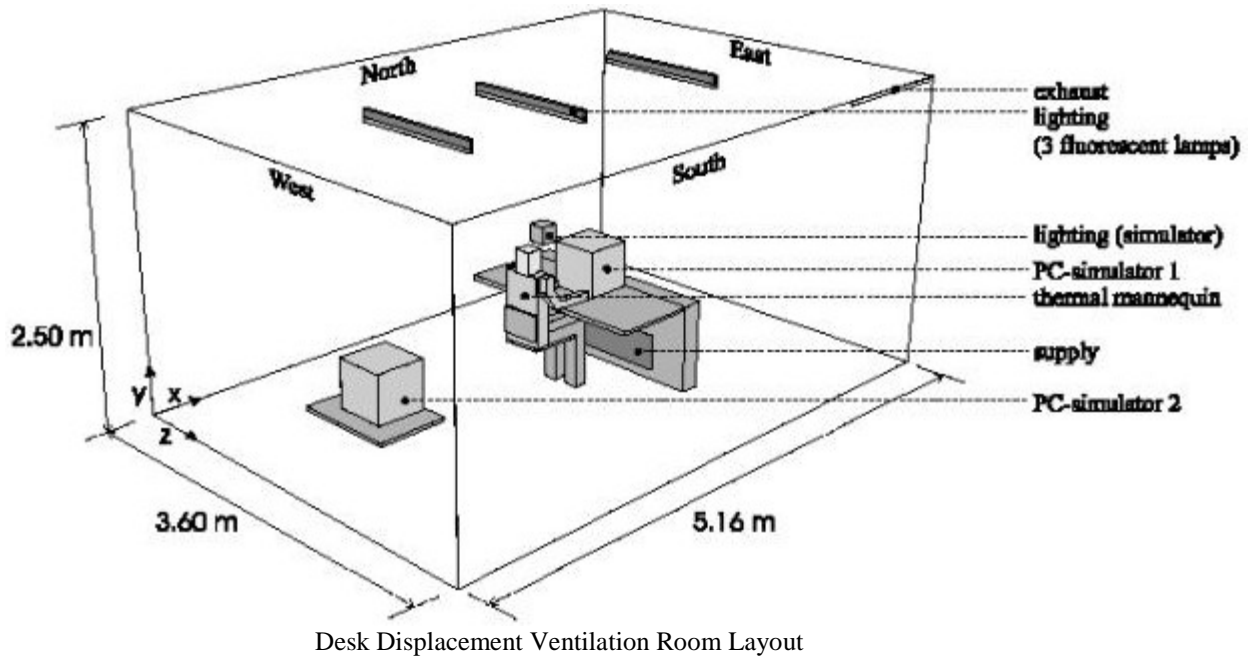


## Displacement Ventilation Experiments

A research paper entitled "The Measurement and Simulation of Indoor Air Flow" written by Marcel Loomans ISBN 90 6814 085 X forms the basis for this section. In his thesis Loomans evaluates the measurement and simulation of the indoor environment. Two measured experiments were undertaken and the data compared to CFD simulated data. The second of these experiments investigates a Desk Displacement Ventilation system (DDV). Using the data for this experiment we have undertaken 3D CFD Tas simulations.

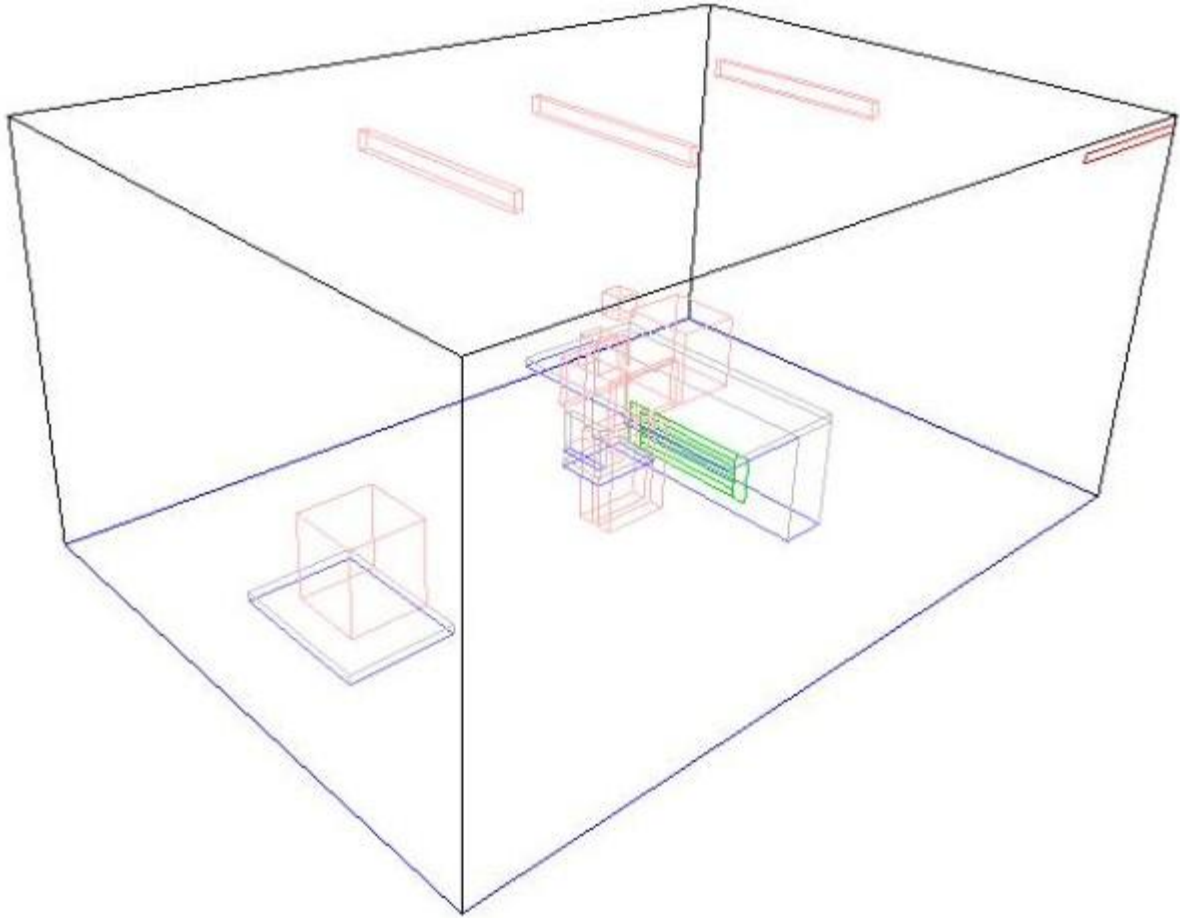
In normal design situations accurate information about the type of heat source or exact distribution are not known. Normal practice in such circumstances is to apply the heat gain at an average rate over a volume at the approximate location. Although the results presented in this section validate the calculation procedures strongly, the main purpose of these simulations was to study the effect of these simplifications. In effect, the goal was not to be as accurate as possible by entering every minute detail of the problem, but to enter the geometry and boundary data as if performing a design simulation. In this way the design process itself becomes the target of the validation exercise. The results would then indicate the practical use of the simulations for basic design work and point out any areas where greater precision in input data was required to achieve the desired accuracy. These cases had different types and sizes of heat gain from internal sources and so made a perfect test for the usual approach of modelling these gains as simple heat 'regions'.



The experiment had been run for eight cases. The boundary conditions were different for each case. The supply rate, supply temperature, mannequin output and power output from computers and lamps were all varied.

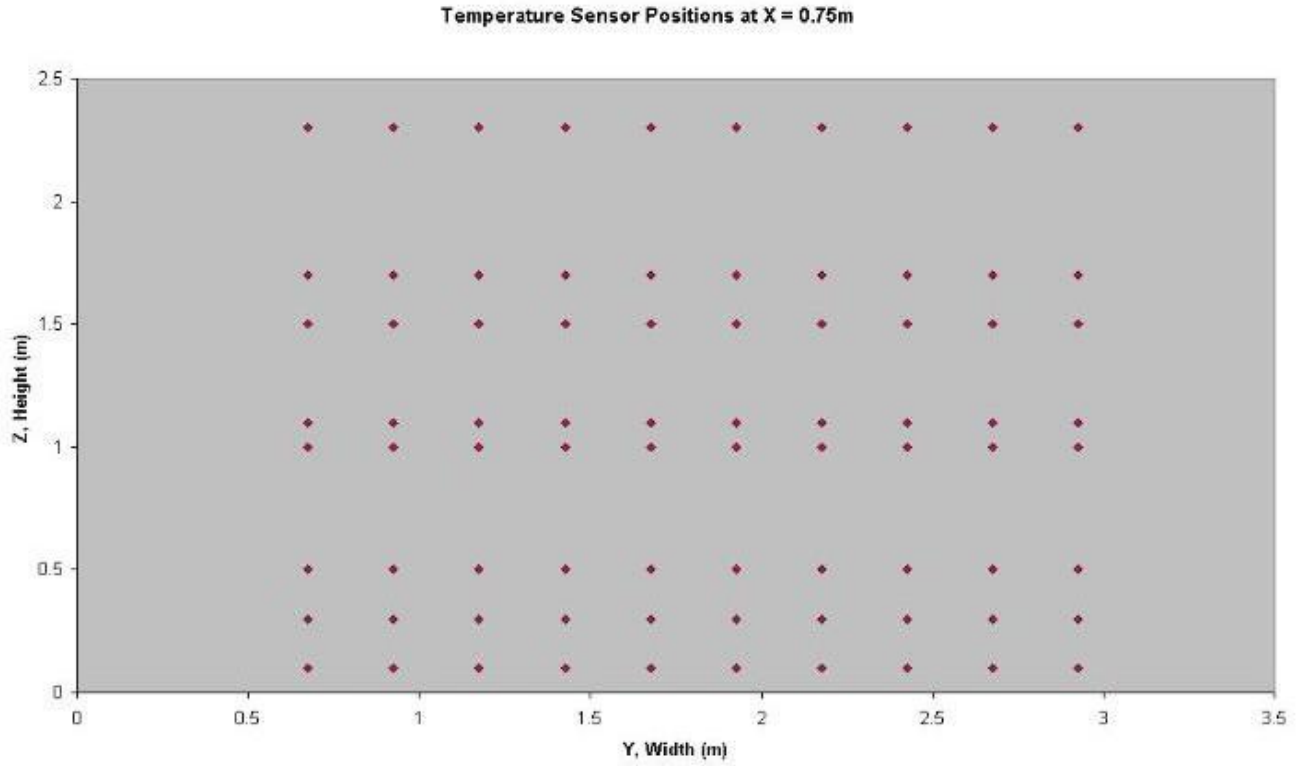
Of particular interest are cases 1a and case 5b. Case 1a has a low supply air flow rate of  $0.010 \text{ m}^3/\text{s}$  and the thermal mannequin provides the only heat gain for the room. In

contrast case 5b has a relatively high supply air flow rate of  $0.029 \text{ m}^3/\text{s}$  and has heat gains from the mannequin and equipment. Case 4a is also simulated and is an intermediary between cases 1a and 5b.



3D CFD Model - Heat Regions, inlets, outlet and blocks are visible inside the Room.

During the experiments temperature measurements are taken using an array of temperature sensors. In all over 500 sensor locations were recorded. The figure below shows the layout for the sensors in the first plane of the room near the west wall looking towards the east. The sensors were numbered from left to right and from the bottom row to the top row, i.e. sensor 1 is in the bottom left corner, sensor 10 is in the bottom right and sensor 80 is at the top right. The next plane of sensors would then follow until all the planes up to the east wall had been covered.



One of Eight Rakes of Temperature Sensors positioned at X = 0.75, 1.5, 2, 2.25, 2.75, 3, 3.75 and 4.5m

### Boundary Conditions

The following table shows the wall temperatures, air supply temperature, supply velocity, mannequin heat output and other heat outputs for each test case. We have simulated cases 1a, 4a and 5b in this comparison. The lower table shows the distribution and radiant proportion for each heat source.

## Boundary conditions

Table F.1. Boundary conditions for the full-scale experiments.

Case	$T_N$ [°C]	$T_S$ [°C]	$T_W$ [°C]	$T_E$ [°C]	$T_F$ [°C]	$T_C$ [°C]	$T_{supply}$ [°C]	$V$ [m <sup>3</sup> /s]	$Q_{tot,m}$ [W]	$Q_{tot,h}$ [W]
1a	23.2	22.8	22.6	22.8	22.5	22.5	20.3	0.010	125	0
1b	23.2	22.8	22.7	22.8	22.5	22.5	22.4	0.010	123	0
2a	23.2	22.7	22.6	22.8	22.3	22.7	19.8	0.029	117	0
2b	23.1	22.8	22.6	22.8	22.3	22.7	19.8	0.029	120	0
3a	23.7	23.1	22.5	22.8	22.7	22.4	20.4	0.010	122	363
3b	23.6	23.1	22.5	22.8	22.7	22.4	20.4	0.010	122	363
4a	23.6	23.0	22.5	22.8	22.5	22.4	19.8	0.019	122	364
4b	23.6	23.0	22.5	22.8	22.5	22.4	19.8	0.019	122	364
5a	23.5	23.0	22.4	22.8	22.4	22.4	19.8	0.029	120	368
5b	23.2	22.7	22.6	22.8	22.2	22.3	19.8	0.029	118	360
6a	23.2	22.8	22.6	22.7	22.2	22.3	19.8	0.047	122	362
6b	23.1	22.8	22.6	22.7	22.2	22.3	19.8	0.047	122	361
7	23.6	23.0	22.4	22.7	22.7	22.4	21.8	0.029	122	366
8a	21.0	20.3	19.3	19.7	20.1	19.3	22.7	0.020	122	361
8b	21.1	20.3	19.4	19.7	20.2	19.3	22.7	0.020	122	360

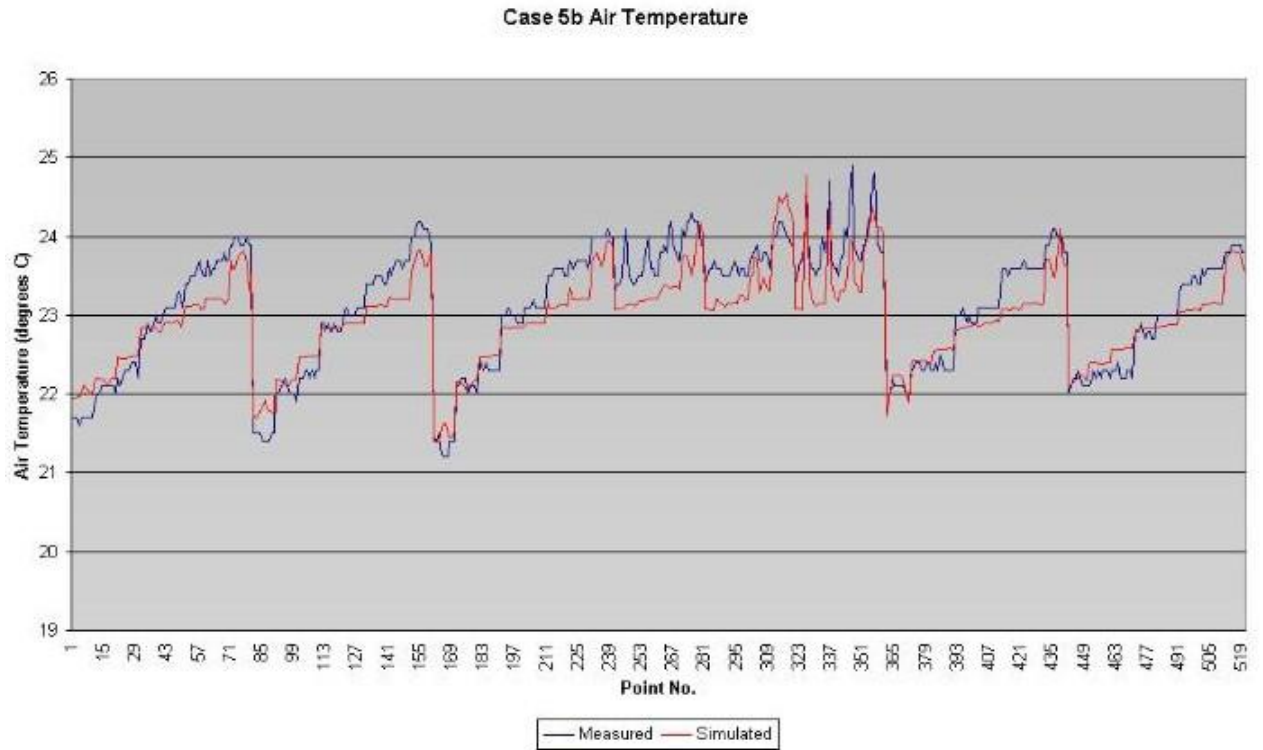
Table F.2. Fraction of total heat input ( $Q_{tot,m}$  and  $Q_{tot,h}$ ) to determine convective heat load ( $Q_c$ ).

sections	head	chest	right-arm	left-arm	right-leg	left-leg
fraction of $Q_{tot,m}$ [-]	0.1	0.13	0.03	0.03	0.10	0.10
heat sources	desk lamp	PC-1	PC-2	Fl. lamp 1	Fl. lamp 2	Fl. lamp 3
fraction of $Q_{tot,h}$ [-]	0.03	0.17	0.17	0.05	0.05	0.05

## Case 5b

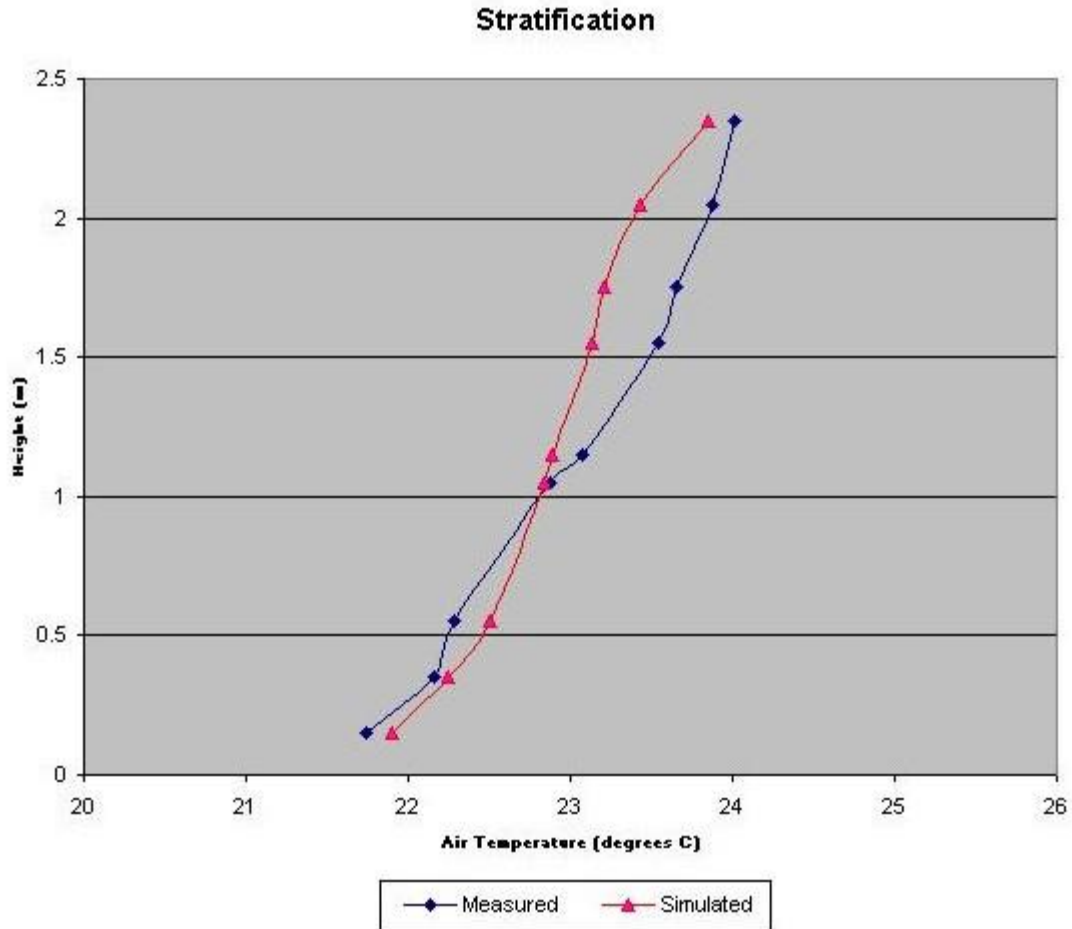
The results for the experimental data are presented as text files and can be downloaded along with the thesis from Marcel Loomans web site <http://www.bwk.tue.nl/fago/AIO/Marcel/thesis.html>. An Automation interface has recently been added to the CFD program that allows external access to the simulation data from any VBA client. Using this automation interface a function in Excel was written which retrieved the air temperature and air speed of a cell for a given sensor coordinate within the domain. The results for Case 5b were opened in excel. Each sensor coordinate was converted and the air temperature function was used to retrieve the corresponding data from the results. The simulated and measured air temperatures are plotted for each sensor point on the chart below.

The Measured and Simulated temperatures are in close agreement. The maximum difference is less than 1 °C. The average difference across all 520 measurements is 0.275°C.



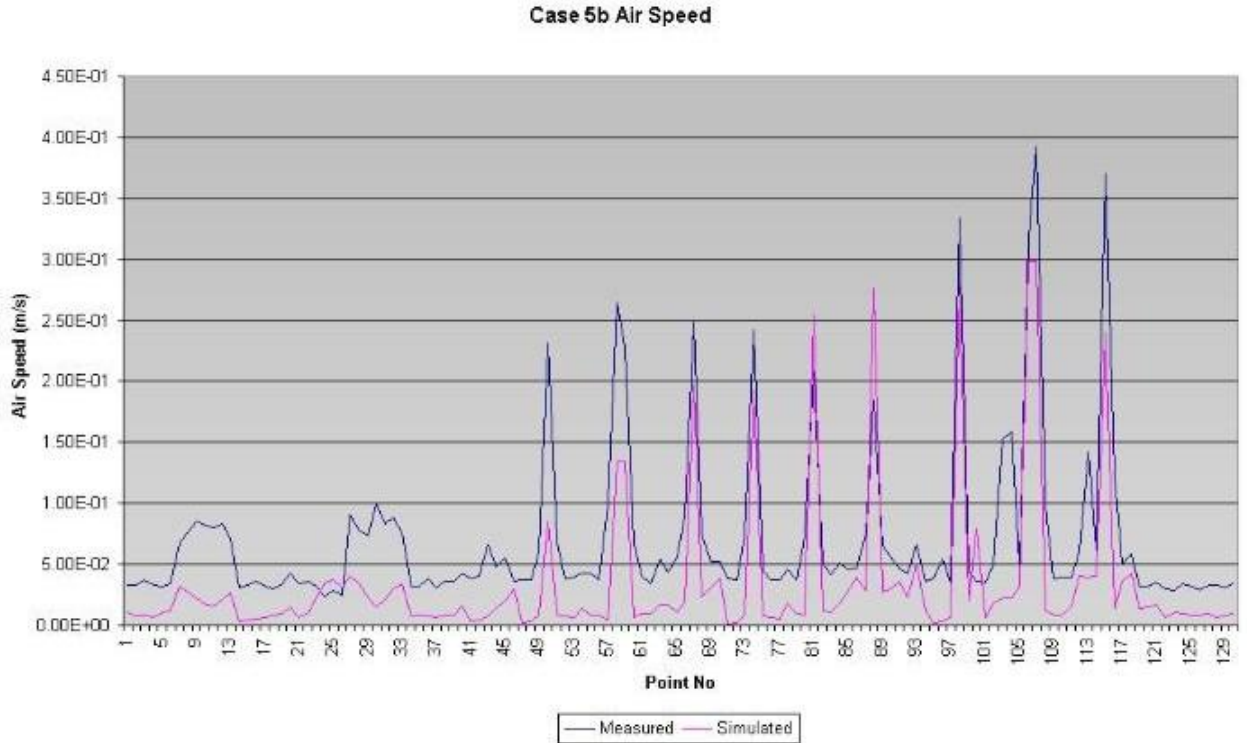
Air Temperature for Measured Experiment and CFD Simulation

The averages of the values taken at the same height are plotted on the graph below. The gradient of the lines gives an indication of how stratified the air is within the room.



Average Air Temperature against Sensor Heights

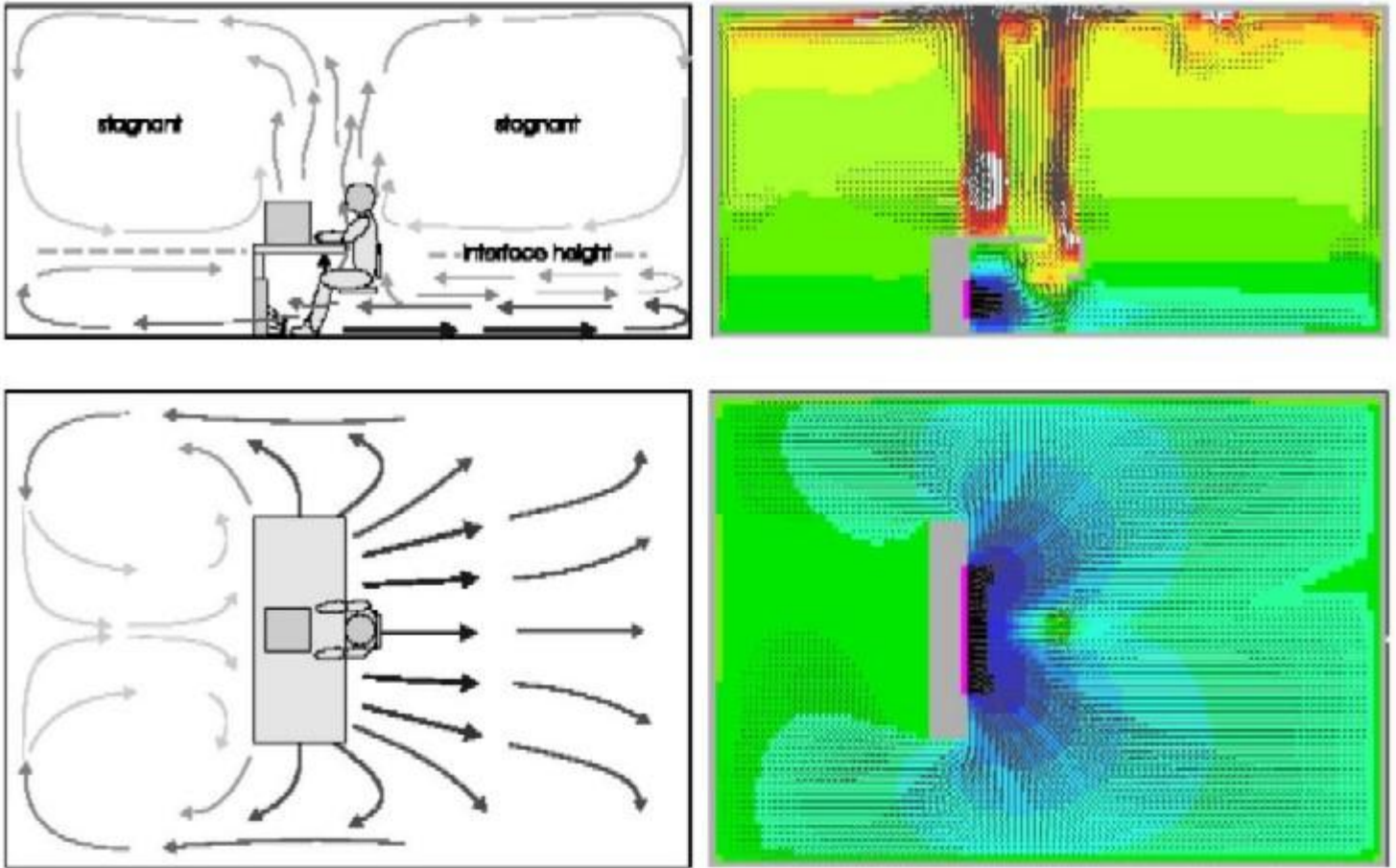
Using the same technique as before the air speeds at each sensor point have been plotted below against their corresponding simulation value. Even though measuring air speeds of less than 0.1 m/s starts to become inaccurate due to limitations in measuring equipment, these results are plotted below as there is a good qualitative agreement in the patterns between the two sets of data. As can be seen, the magnitude and location of the hot plumes has generally been predicted well.



Air Speed for Measured Experiment and CFD Simulation

The figure below (left hand side) shows a sketch of the flow pattern produced using video recordings of the diffusion of smoke. On the right are the corresponding 3D CFD plots. The different colours represent temperature bands and the velocity vectors illustrate the flow pattern. There is a very good qualitative match between the two diagrams. It is very pleasing to note that the CFD simulation has captured the change in interface height, the velocity vectors clearly showing the air separating from the left and right wall at different heights.





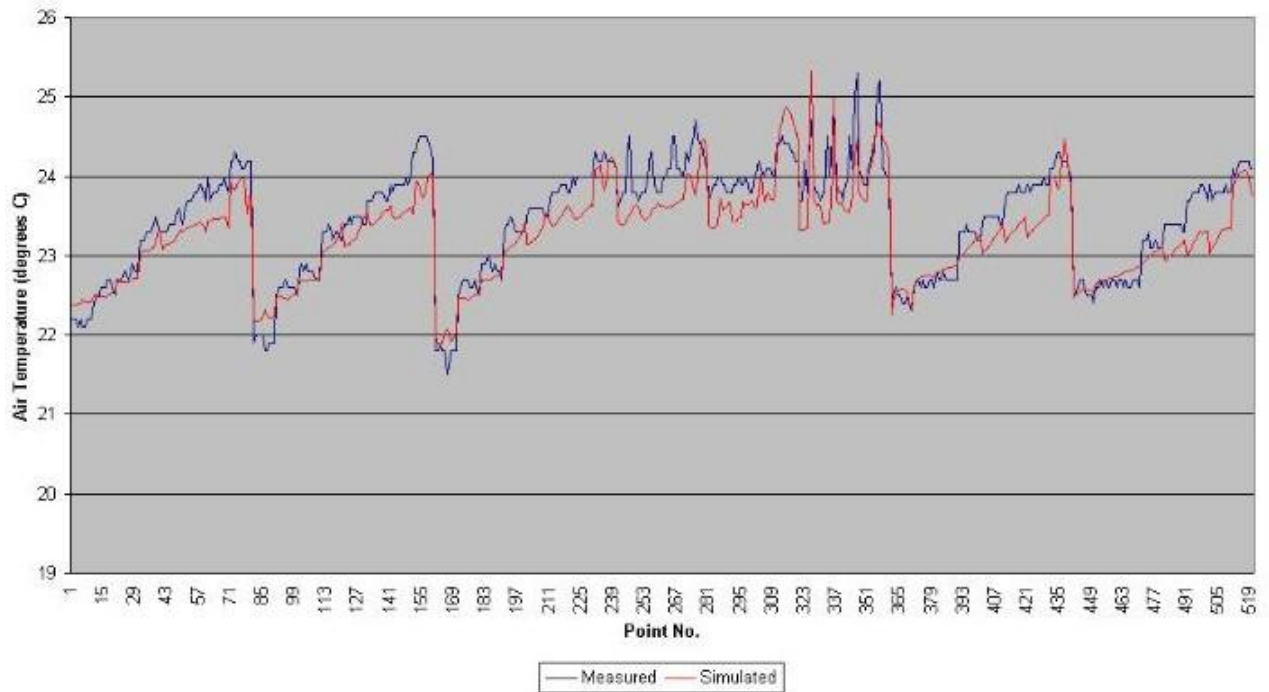
Flow features of the displacement ventilation flow pattern in the central plane of the room and at a horizontal plane at ~0.1m above floor level.

### Case 4a

The Measured and Simulated temperatures are in close agreement. The maximum difference across all 520 sensor locations was just over 1 °C. The mean of the differences between the two methods was 0.275°C.



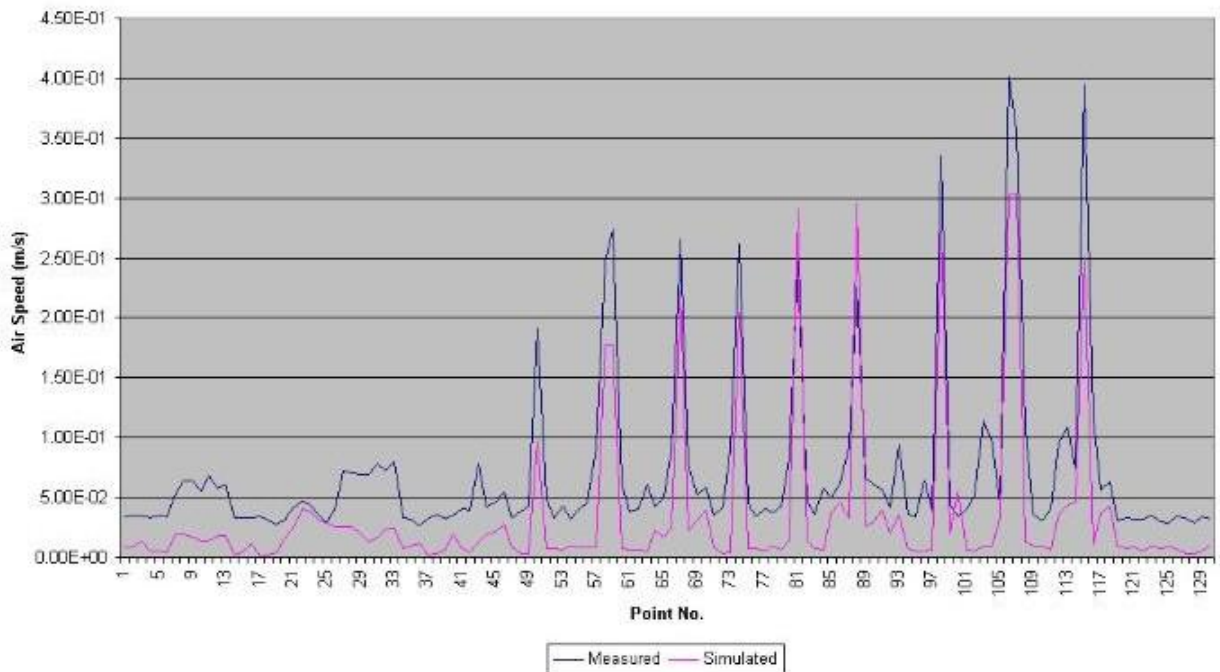
Case 4a Air Temperature



Air Temperature for Measured Experiment and CFD Simulation

The air speed results are shown below. As in the previous case 5b the plumes are captured well. Once again only the plume over the mannequin seems a little slow probably indicating a problem with its exact location or intensity.

Case 4a Air Speed



Finally in this section the following table compares some general experimental observations with the simulated equivalents in blue. Overall the agreement is

extremely good. The lower velocity at location 2 is probably due to the slight difference in plume shape. A very small change in plume shape or direction can have very large effects on the measured air speeds.

<i>Case</i>	$T_{supply}$ [°C]	$T_{exhaust}$ [°C]	$Q_{c,v}$ [W]	$\xi$ [-]	$\bar{T}_{0.1m}^*$ [°C]	$dT/dy$ [°C/m]	$\bar{u}_{max(1)}$ [m/s]	$\bar{u}_{max(2)}$ [m/s]
4a	19.8	23.8	93	0.37	22.2	1.00	0.27	0.25
4a (CFD)	19.8	23.49	86	0.35	22.3	0.82	0.19	0.26

$Q_{c,v}$  : Heat load removed from the room by ventilation.

$\xi$  : Air cooling fraction ( $= Q_{c,v} / Q_{tot,c}$ ).

$\bar{T}_{0.1m}^*$  : Temperature at 0.1 m height averaged over all measured temperatures at 0.1m height.

$dT/dy$  : Vertical temperature gradient (0.1 m - 2.3 m height) [averaged value over six positions].

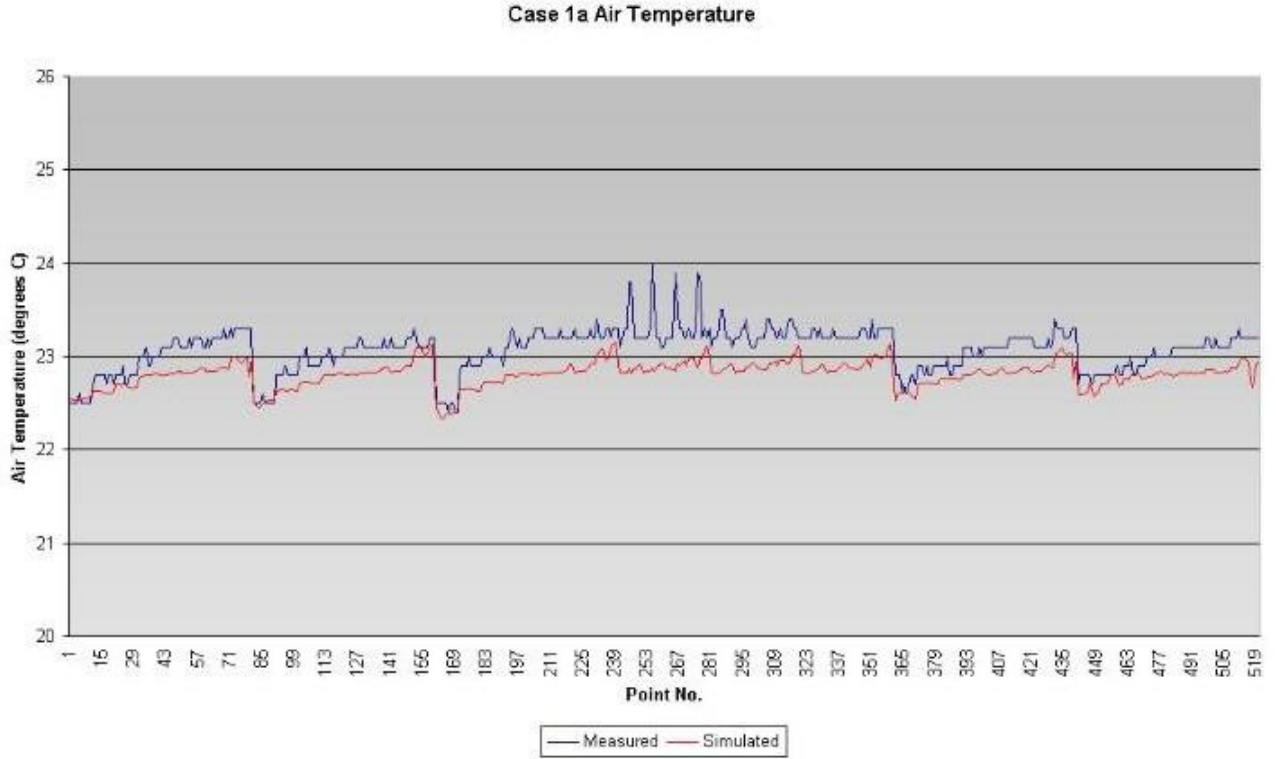
$\bar{u}_{max(1)}$  : Velocity in plume thermal mannequin at 2.0 m height ( $x = 2.25$  m).

$\bar{u}_{max(2)}$  : Velocity in plume thermal mannequin at 2.0 m height ( $x = 2.40$  m).

Characteristics of the measurement (4a) and simulation (4a(CFD)) results

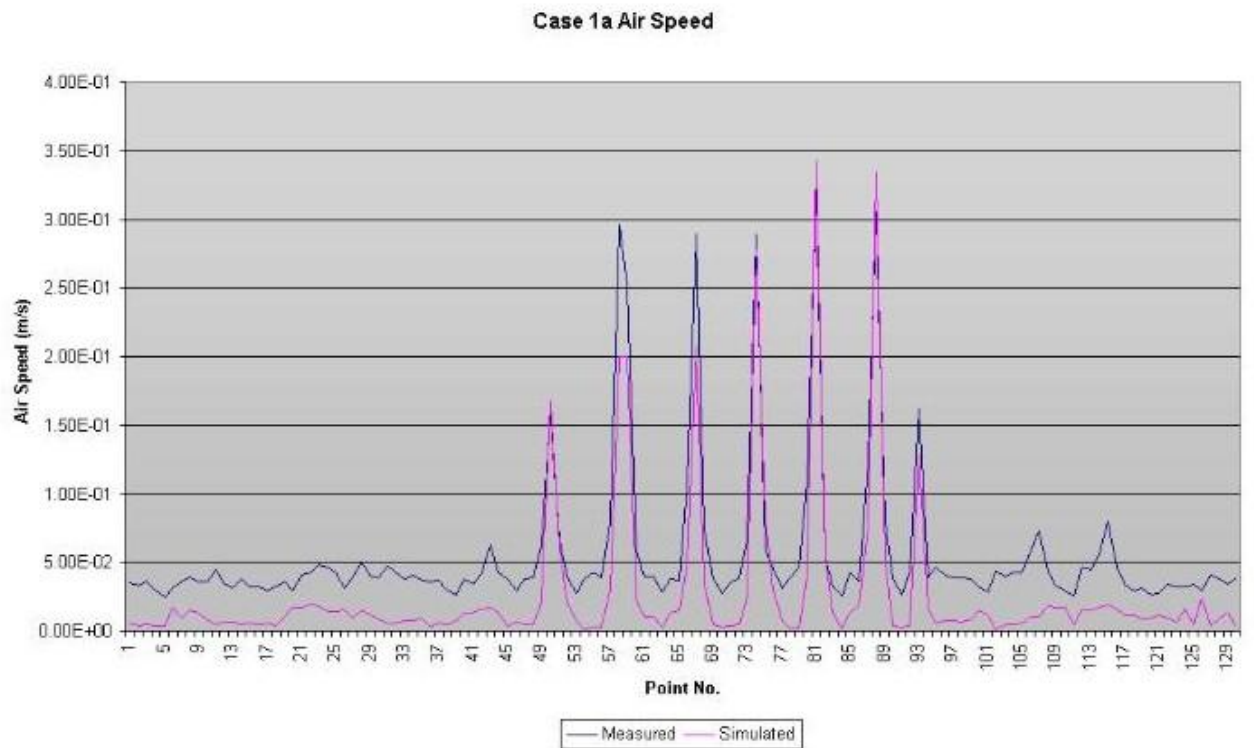
## Case 1a

The Measured and Simulated temperatures are once again in close agreement. The peak difference was 1.13°C and the average difference between the two methods was 0.263°C.



Air Temperature for Measured Experiment and CFD Simulation

The air speed results below are particularly impressive indicating that the plume above the mannequin has been captured very well in this particular case.



Air Speed for Measured Experiment and CFD Simulation

## Conclusion

From studying these results the main effects of using heat gain regions appear to be

- a slight reduction in overall plume speed and shape
- a similar reduction in peak temperature and location.

Both these effects can be explained by the fact that the real heat sources were solid objects and the heat was transferred only from their surfaces into the neighbouring volume. In the simulation the heat is spread over the entire volume of the object and not into the small volume adjacent to each surface. This 'dilution' of the heat gain would lead to a reduction in temperature difference and consequently speed, exactly as seen in the simulation results. In the future we hope to conduct further runs where the heat source is modelled as surface heat flux and also runs where the heat source is distributed as a smaller number of higher intensity gains. However, it seems unlikely that either approach will provide a large improvement given the good accuracy already obtained.

These results show it is perfectly acceptable to model quite diverse types of heat source as simple heat gain 'regions' with relatively little loss in accuracy. In fact, the simulations in all these cases produce results that are more than accurate enough for design purposes.