

## Experimental Validation – 3D CFD

### The Pit Project

EDSL and partners\* commissioned a series of experiments to be undertaken to investigate the interaction between heaters and a coupled room. The experiments took place at the EMC test site at Cranfield in 1996 and 1997.

\*The Building Research Establishment Ltd. (BRE), Electricité de France (EDF), and The Energy Monitoring Company Ltd. (EMC) <http://users.powernet.co.uk/emc>

During the testing instrumentation placed in the walls and throughout the test room (163 sensors in all) collected data at five minute intervals. Two sparge pipes on opposite sides of the room run down from the ceiling to ventilate the space. Outside air is pumped into one of the pipes and out of the other at a rate of one air change per hour. The south facing side of the room is exposed to the outside and has a double glazed window. Two different types of heater were used in the experiments varying both the heater position and power output.

In this validation exercise the data used is from two of the experiments conducted during the pit project. The first positions the convective heater under the window along the south face of the test room (Experiment 1) and the second positions the heater along the opposite north face of the room (Experiment 2).



Pit Test Room



Wall Temperature Sensor



Sparge Pipe Inlet and Outlet



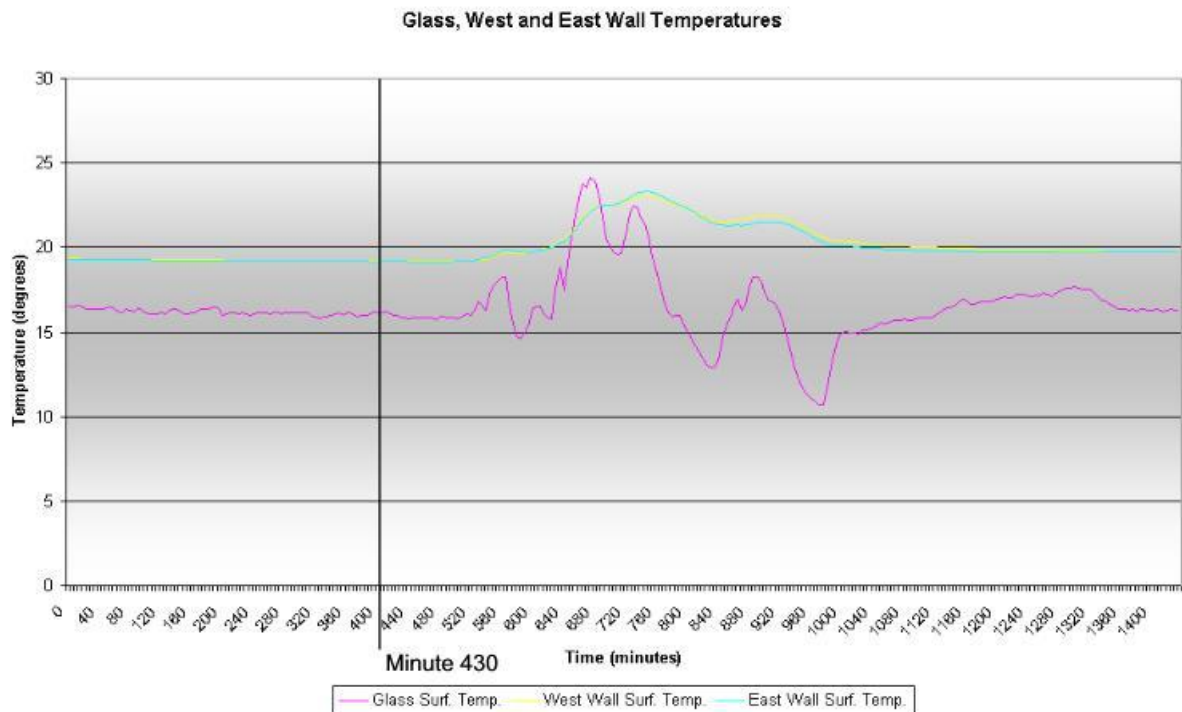
Located in the roof of the test room - Data Collection, Recording and Control.



View inside test room

### Setting Boundary Conditions

Below is a graph to illustrate the method used to select a suitable time in the recorded dataset. All the boundary conditions required for the CFD Simulation (i.e. surface temperatures, heater power etc.) are shown to be steady for a period of time at minute 430 (only three conditions plotted here). This shows that the thermal processes of the room are very close to steady state at this time. To validate the CFD calculations we can now compare the recorded room air temperature data with the simulated air temperatures from a steady state CFD simulation.

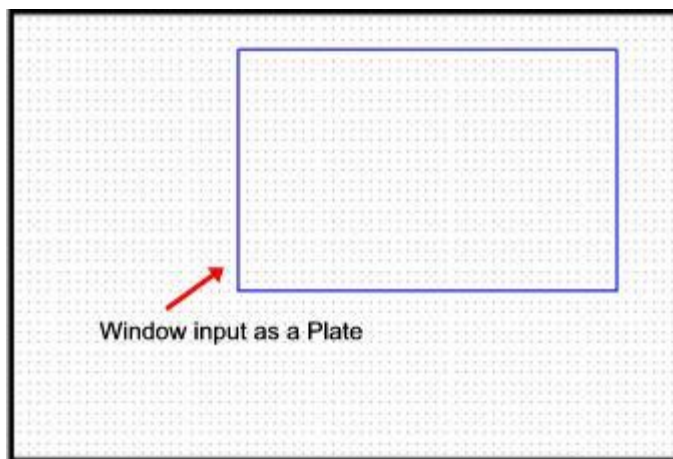


Select Minute 430 for Boundary Condition Values

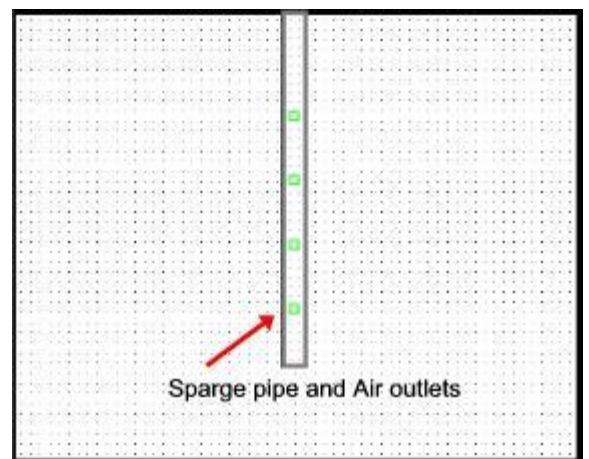
### Creating the CFD Model

The first stage in creating a 3D CFD Model is to choose an appropriate cell size. For this model each cell represents a cube 0.05 x 0.05 x 0.05m. The geometry of the room is entered on the grid. Sparge pipes, inlets, outlets, isothermal plates and heating regions are added to complete the model.

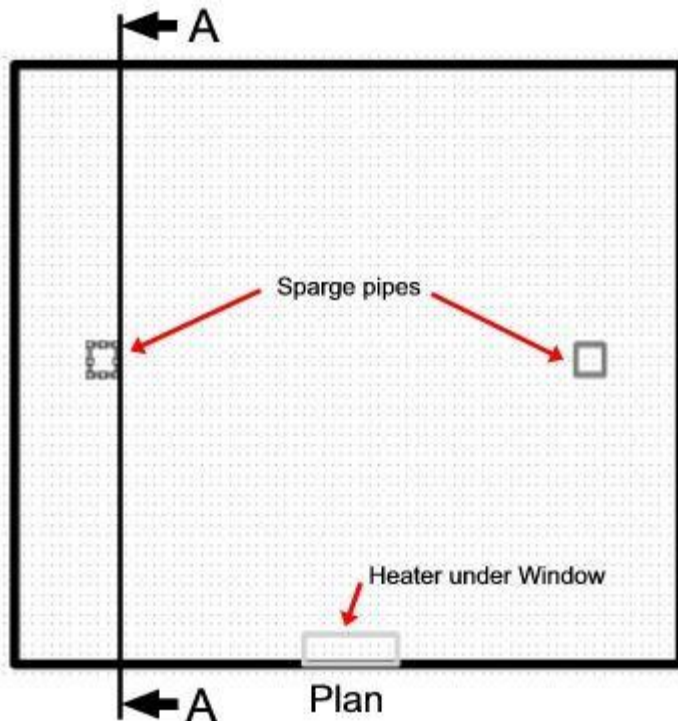
The boundary conditions are then assigned to the model. For the floor, ceiling and walls the average temperature reading for each surface is calculated from the recorded data. The flow rate for the inlets and outlets is also set along with the inlet air temperature. The sensible heat output for the convective heater is entered as the magnitude for the heat source region.



South Elevation



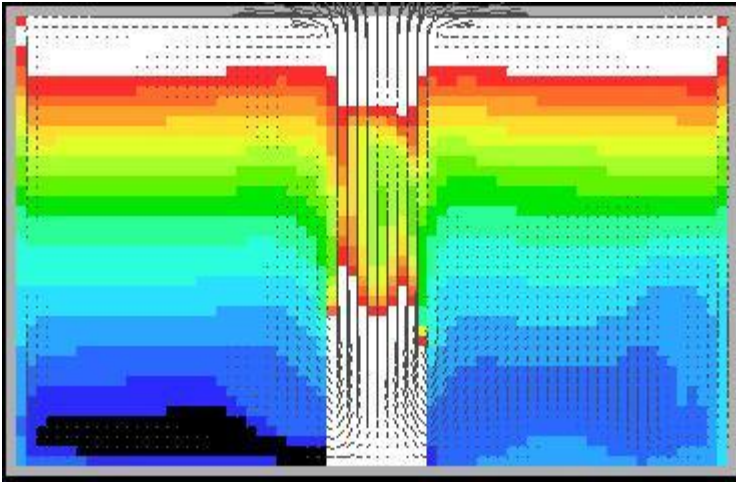
Section A-A



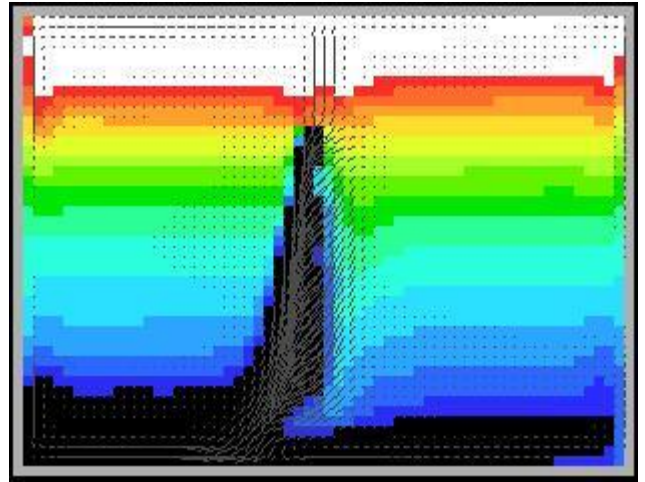
## Running the Simulation

The simulation is run until the air temperature distribution remains stable and steady state has been reached (the case where the heater is under the window is shown here). By changing the position of the slices and using the temperature probe the values for both temperature and velocity can be shown for any position within the room.

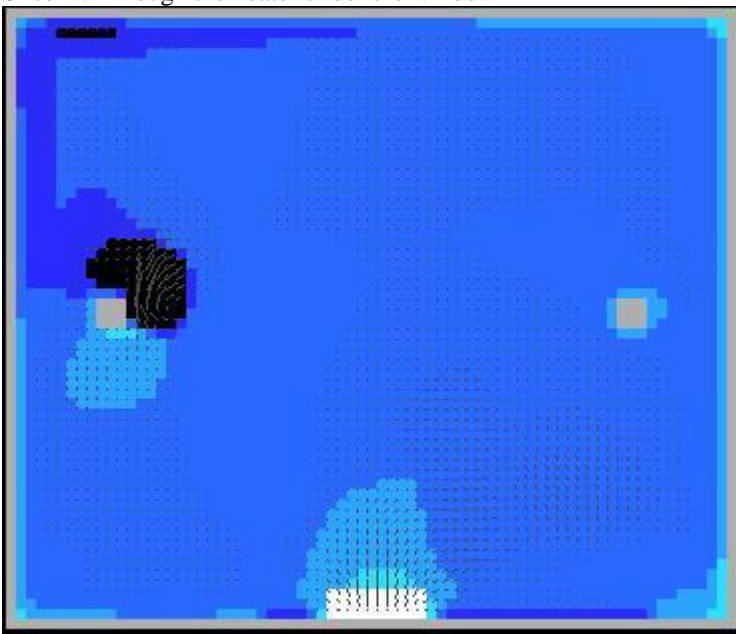
Using the information on the position of the air temperature sensors the simulated temperature for each of the sensors is recorded.



Slice Y: Through the heater under the window



Slice X: Section through west sparge pipe



Slice Z: Section through sparge pipes and Heater

## Sensor Positions

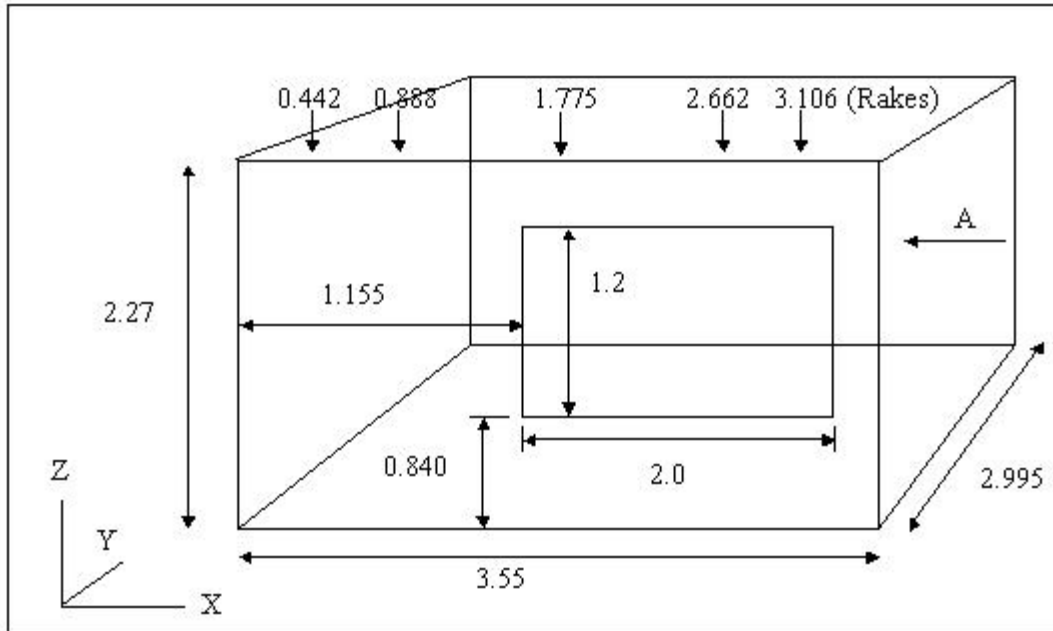


Figure 1. Test cell dimensions with positions of sensor rake, all dimensions in m.

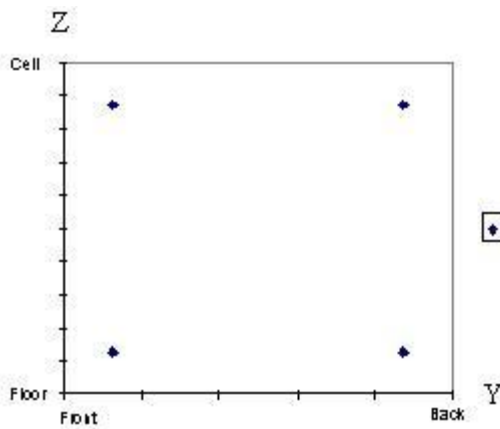


Figure 2. Sensors on rakes at 0.442m&3.106m

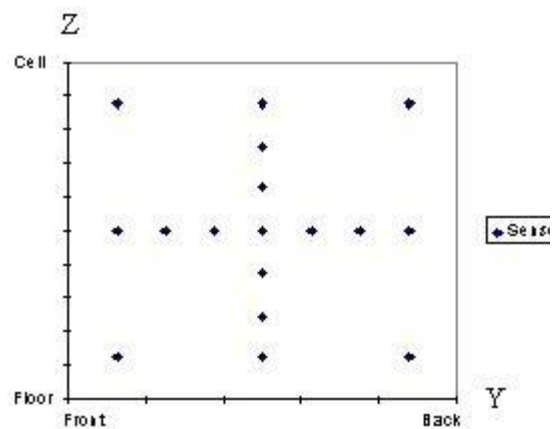


Figure 3. Sensors on rakes at 1.775m

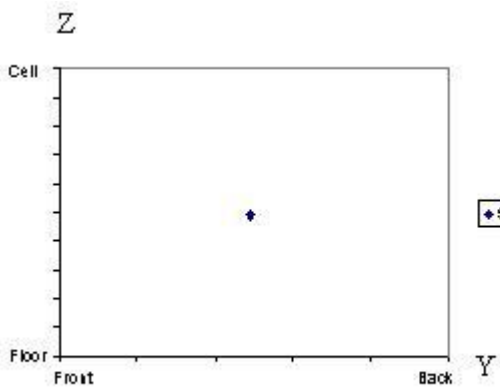


Figure 4. Sensors on rakes at 0.888m&2.662

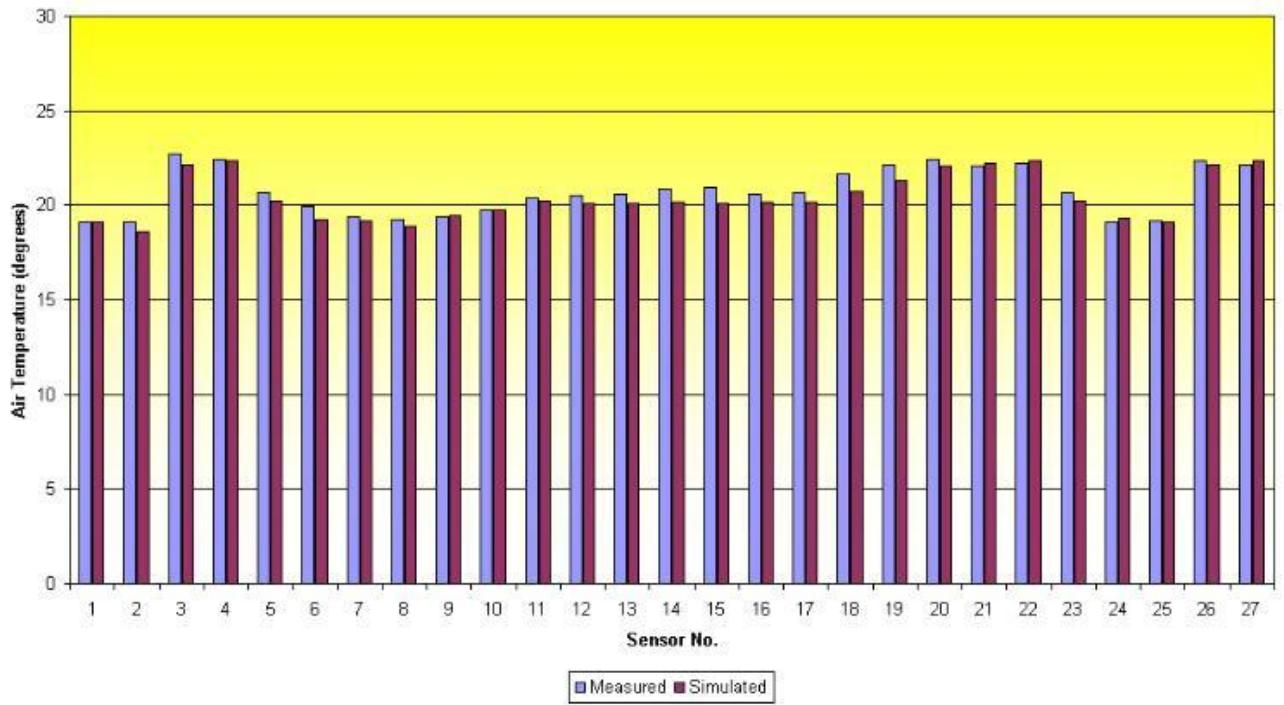
Figure 1 - 4 show the positions of the air temperature sensors in the test room.

NB. Figures 2 and 4 show sensor positions for two rakes.

In all, 27 air temperature sensors are scattered through the room. There was an additional velocity sensor positioned centrally below the window on the south wall.

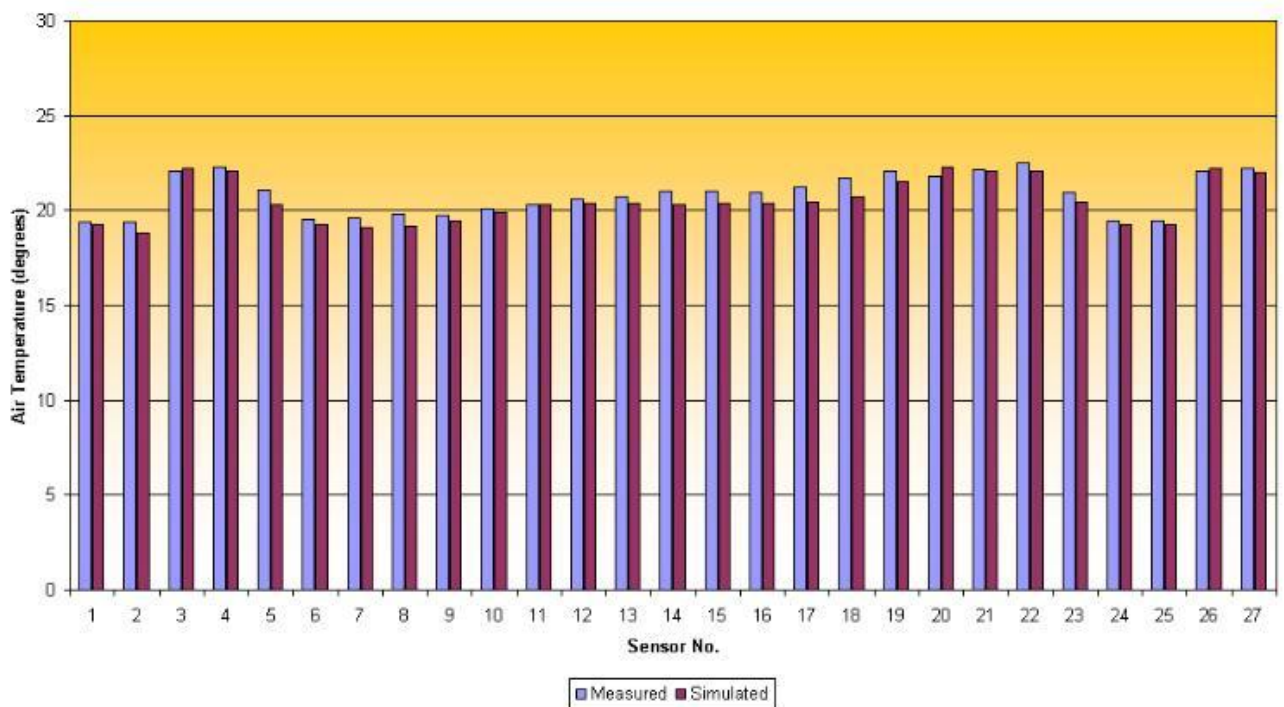
## Experiment 1

Measured and Simulated Air Temperatures - Heater under Window



## Experiment 2

Measured and Simulated Air Temperatures - Heater along North side



The simulated and measured results are in very good agreement. The Y-axis scale has been chosen to represent the range in temperatures that existed within the test cell. The 1 air change of outside air was  $-0.94^{\circ}\text{C}$  for Experiment 1 and it was  $3.02^{\circ}\text{C}$  for Experiment 2, so zero has been chosen as the lowest value on the scale. The air leaving the heater was approximately  $30^{\circ}\text{C}$  so this has been used as the upper value. The simulated velocity for Experiment 1 was  $0.48\text{m/s}$  compared to a measured value of  $0.52\text{m/s}$ , again, the values agree very well.

Experiment 2 with the heater against the north wall measured air speeds too low for the tolerance of the device. The simulated results were also tiny. This does not represent such a strong validation because experimental values at such low speeds are questionable.

The simulated results are consistently a little low through the centre of the cell. We believe this a result of the re-laminarization of the air flow which the current turbulence model does not capture as well as we would like. This is an aspect of the software we are still studying to try and obtain even greater accuracy.

## **Next stages in validation**

We will be publishing the results of our dynamic validation runs soon. These are tests where the time varying boundary conditions from the experiments drive a minute by minute dynamic CFD simulation and the outputs are compared not only through the internal space of the test cell but also through time. As far as we know this will be the first time a simulation of the dynamic response of the air mass inside a room will have been validated against an experiment.

To show the power of the Building Designer and Dynamic Ambiens combination we will be posting a comparison of the complete dynamic simulation of the PIT experiment set. In this exercise Building Designer predicts the surface temperatures and other boundary information that is then used in the dynamic CFD run. These combined results will be compared to the measurements. This will validate the typical design procedure where weather conditions, building fabric etc. are the inputs for dynamic simulation, which leads to surface temperatures etc. to be used in the CFD calculation. As we already have good agreement for Building Designer and Ambiens in their individual comparisons we expect that the combined runs will also show very good agreement with the measured results. This shows that dynamic simulation in combination with CFD allows an engineer to predict the temperature and air movement, at any point within a space of a building, from only simple design data.