CFD validation of the thermal comfort in a room using draft rates

Authors: Mika Ruponen[1] and John A. Tinker[1]

[1]School of Civil Engineering, University of Leeds, UK

Corresponding email: mika.ruponen@halton.com

SUMMARY

Air temperature and velocity are the two main factors affecting the thermal comfort indoors. These two values can be easily obtained using computational fluid dynamic (CFD) simulations together with the turbulence kinetic energy value. This paper evaluates methods of calculating thermal comfort indices using CFD. Simulated results are compared against experimental data measured in a purpose build full-scale model room. The results show that CFD data can reliably predict thermal comfort values.

INTRODUCTION

Computational fluid dynamics (CFD) has become a popular tool for evaluating air distributions in a buildings and it is becoming common practice to simulate different designs alternatives before building a full or reduced-scale physical model. Quite often CFD is used alone to evaluate thermal comfort conditions in a room at design stage and when this is the case, it becomes important that both the user and method are validated against some form of measured data.

The main objective of this work is to validate how closely simulated results represent measured thermal comfort conditions in a room. Experimental data for CFD validation purposes is readily available [1-3]. However, as mentioned earlier, CFD is often used for initial design instead of building a physical model. In many cases the final design of the physical model can be very different from the earlier simulated design.

Air velocities are traditionally measured using omni-directional hot-sphere anemometers which record an average air speed (V). This averaged air speed reading consists of a time averaged unidirectional velocity component as well as an unidirectional turbulent fluctuation. The comparable result from a CFD simulation is a directional velocity vector (Vv) which includes a turbulent kinetic energy component. This has being addressed by Koskela et al [4] and a method has been developed to correct the vector value to a simulated speed value (Vo). The directional results obtained from CFD simulations are being reported to have smaller values than the omni-directional results obtained using a hot-sphere anemometer. Thermal indices such as the Draft Rate (DR) defined in ISO 1994 [5] are based on omni-directional results where the effect of the proposed correction for velocity on DR are evaluated.

METHOD

Test room

The test room had a floor area of 2.83m x 4.725m and a height of 3m and was surrounded by a cavity in which the temperature could be controlled. The room had insulated walls, ceiling and floor to reduce the heat flow through the surfaces. A bulkhead 300mm high and 600mm deep was built inside the room to accommodate the supply and extract ductwork. An air handling unit (AHU) was located adjacent to the test room which provided a temperature and volume controlled supply of air to the room.

Supply and exhaust of air into and from the test room was provided via two 100mm ϕ ducts located in the bulkhead as shown in figure 1. Two dummies [6] also as shown in figure 1, were used to provide a potentiometer controlled heat load into the space. The dummies were absent in case 1.

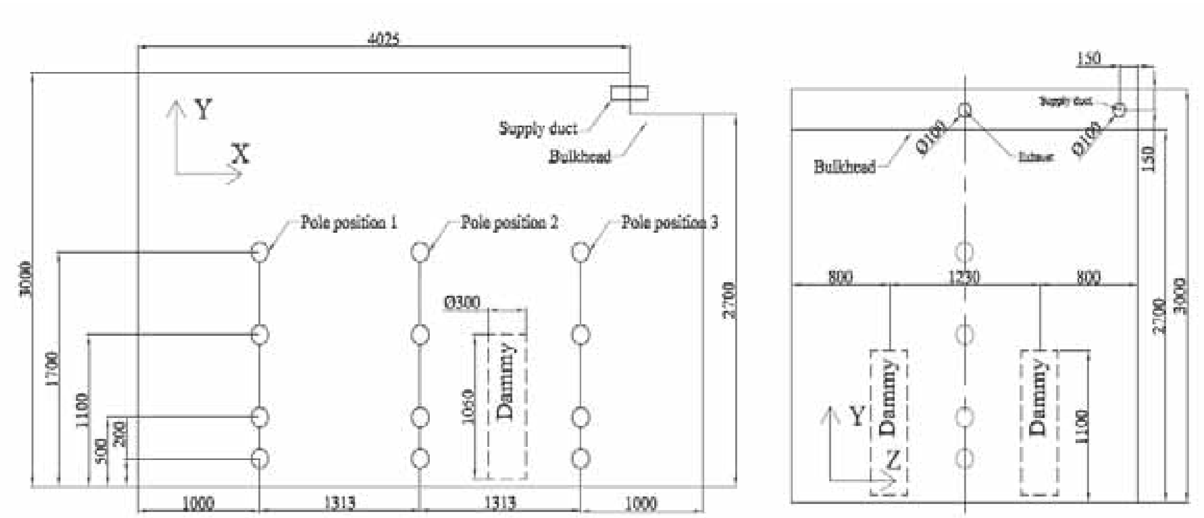

Figure 1. Test room plan and dimensions from Z and X-directions

Measurement equipment

The supply and extract volume flow rate was measured using calibrated MSD flow measurement units whose accuracy was ±5%. Temperatures were measured using PT 100 temperature probes that were calibrated to an accuracy of ±0.3K over the temperature range 15°C to 35°C. Omni-directional hot-sphere anemometers were used at various locations to simultaneously record air velocity to an accuracy of ±0.02m/s. The velocity and temperature sensors were fixed to a pole. Each pole had 4 sensors to measure air velocity and 4 for temperature (figure 2). The sensors were fixed 200mm away from the pole to reduce any effect the pole may have on the flow. Light-weigh paper strips were placed immediately below the sensors to visualise the air flow pattern in the room in addition to using a smoke tracer system. The heat load from the dummies was controlled using potentiometers and the power to them recorded using current meters.

Figure 2. Velocity and temperature measurement equipment

Test Procedure

Two different experimental test cases were used to study the conditions and flow fields in the room as shown in Table 1.

Table 1. Specification of the test cases

Case	Supply air (l/s)	Supply air temperature (°C)	Air extraction rate (l/s)	Dummy heat load (W)
1	50	23	50	0
2	50	19.2	50	240

In both tests the cavity air temperature was controlled at 23°C to minimize the heat exchange between the cavity space and the test room. The supply air temperature and volume were controlled to the specified rates and prior to any temperature and velocity measurements being recorded, the room was allowed to stabilize. A stable condition was assumed when the temperature difference across the room was less than ±0.5°C. In case 2, the desired room temperature under a heat load was controlled by reducing the supply air temperature.

Once the room had stabilised the measurement poles were placed in the room at the predefined locations as shown in figure 1. The test room was allowed to stabilize for a further 3 minutes before measurement was started. Each measurement period lasted for 3 minutes with recordings being taken at 1 second intervals and averaged. Each measurement was repeated 5 times to ensure time independent results.

Computational model

Several commercial CFD codes are available which can be used to simulate room air conditions. In this work ANSYS CFX 10 [7] was used. The continuity, momentum and energy equation were used to calculate the 3-dimentional flow field and heat transfer. The finite control-volume method was implemented for the spatial discretisation of the domain.

The geometry and boundary conditions used in the simulations represented the test room. The vertical surfaces of the dummies were defined as surfaces with a heat flux. The K-ε turbulence model was initially used because it has been regarded as industry standard for many years and its robustness and reliability has being reported in many studies [8-10]. One of its weaknesses is its ability to predict convection flows. In the most types of indoor spaces, convection flows are present and the STT turbulence model has a reputation for predicting these realistically [11] although comparisons are difficult to find. In this work CFD simulations have been made using both the K-ε and STT turbulence models and results reported.

Method used for calculating Draft Rates

The basic equation to calculate the DR is presented in ISO 7730 [5]. Turbulence intensity cannot be obtained directly from CFD results but a method to calculate turbulence intensity is presented by Koskela et al [4].

RESULTS

The velocity and temperature results were shown not to be time dependent. The variation between the results taken over five 3 minute measurement periods was less than the accuracy of the equipment used.

The average velocities recorded at the 12 measurement points are shown in Table 2 and as can be seen, simulated and experimental results show fairly good agreement. The SST simulated values show better agreement in the non-isothermal case 2. The values reported are average values taken over 5 measurements.

Table 2. Average velocity values for experiments and simulated cases

	Experiment	Simulation SST		Simulation K-E	
Case	V	Vv (m/s)	Vo (m/s)	Vv (m/s)	Vo (m/s)
1	0.20	0.16	0.20	0.18	0.23
2	0.21	0.20	0.23	0.18	0.20

The simulated and experimental velocity readings for the isothermal test (case 1) are shown in figure 3. Measurements recorded at different locations show a similar trend with regions of high and low velocities agreeing with simulated results. The biggest difference being equal to, or less than 0.1 m/s.

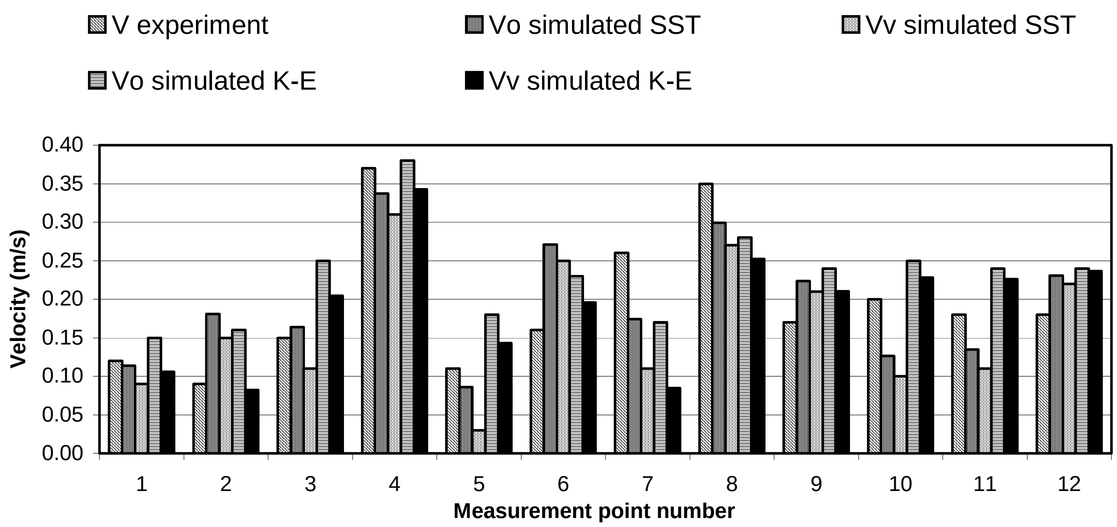

Figure 3. Velocity readings for experimental and simulated data for the isothermal test case 1

The experimental and velocity readings for the non-isothermal test (case 2) are shown in figure 4. Again in this case, experimental and simulated results agree closely but the difference between the results using the two different turbulence models is becoming evident. Both turbulence models seem to have difficulties accurately simulating high and low velocities.

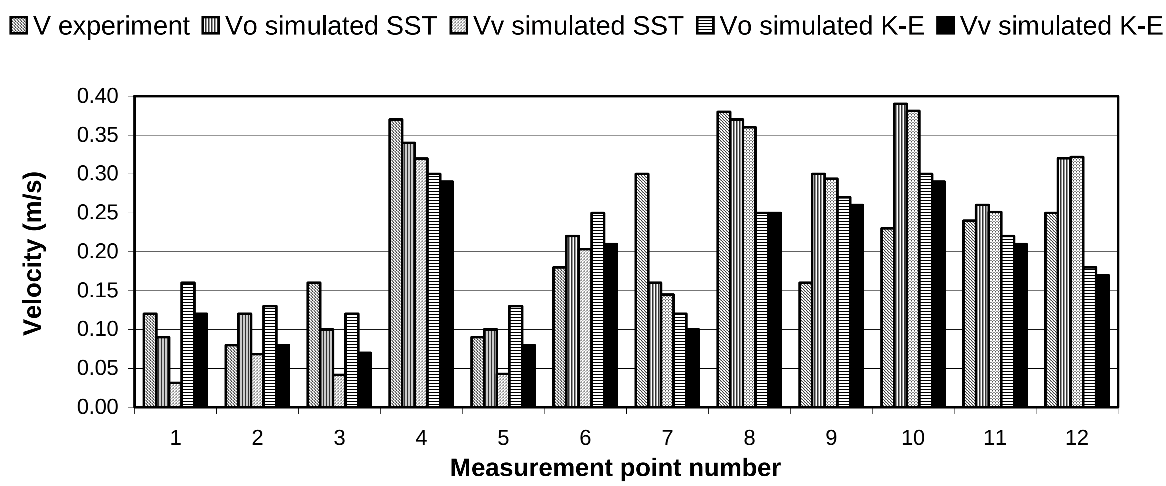

Figure 4. Velocity readings for experimental and simulated data for non-isothermal test case 2

The average draft rate (DR) results shown in Table 3 were obtained using the method reported by Koskela et al [4]. The DR values are important because in most cases, thermal comfort predictions are based on this value.

Table 3. Average Draft Rate percentage values obtained from the measured and simulated values.

Experiment	SST		K-E	
V	DR Vv	DR Vo	DR Vv	DR Vo
18	15	18	15	17

As can be seen from the table, the percentage difference between all the average DR values is small. The omni-directional simulated values agree closely with the measured data. The vector values are lower as would be expected since the correction is only applied to the higher values. Figure 5 shows the measured and simulated draft rates at various locations in the room and as can be seen, the results are in line with earlier results.

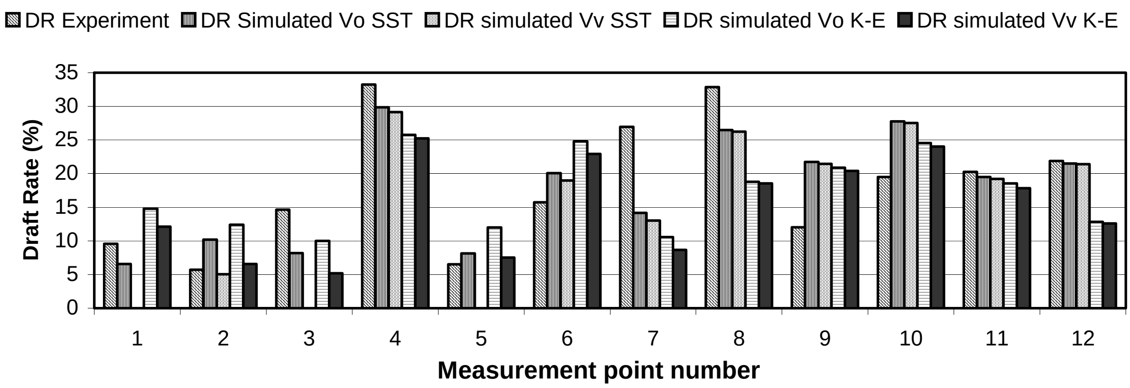

Figure 5. Draft Rate values for the experimental and simulated data

DISCUSSION

The purpose of the work was to study the capability of CFD to predict the comfort conditions in a room under different air flow scenarios. Experimental work was carried out in a full-scale test room and the same geometry and boundary conditions then used in a computational model. The experimental results are in good agreement simulated values although some differences are evident.

The results show that CFD is a useful tool to predict comfort conditions at design stage. In this study a simple room geometry was used which produced a complicated flow scenario. Both turbulence models were able to simulate the flow reliably but the performance of the SST model was better.

All new room designs have to guarantee specified levels of comfort. Common target levels of either DR 15 or 20 are specified for new designs. In addition, a maximum velocity can also be specified. The results of this study were compared with the DR 15 and DR 20 threshold values. Table 4 shows the number of points that are above these defined threshold values. CFD simulations using the SST model seem to agree slightly better with the experimental results although the differences are very small. Omni-directional CFD values seem to have very little effect on the DR.

Table 4. Measurement points above threshold values

Treshold	Experiment	Simulation SST		Simulation K-E	
DR	V	Vv (m/s)	Vo (m/s)	Vv (m/s)	Vo (m/s)
15	7	7	7	6	6
20	5	5	5	4	3

Great care has to be taken when results are presented and comparison made. The methods used to obtain experimental results must clearly stated to clarify whether the results are vector values or omni-directional values. In the present study there was little notable difference on the DR but for validation, velocity and temperature correction is highly recommended.

CONCLUSIONS

- The work showed that CFD can be reliably used to evaluate the thermal comfort provision in a new design.
- CFD simulations using a K-ε turbulence model have slightly better agreement with experimental velocities particularly in isothermal test cases. In non isothermal cases, SST turbulence models perform better.
- Correcting simulated velocity results to an equivalent omni-directional value has negligible effect on any subsequent DR evaluation. Use of a correction is however recommended when experimentally measured omni-directional velocity results are used for validation purposes.

ACKNOWLEDGEMENT

The financial and other support of Halton Oy is acknowledged and this enabled the work to be carried out.

REFERENCES

1. Lemaire, A. D.1992, Room Air and Contaminant Flow- Evaluation of computational methods, Subtask-1 Summary report, IEA ANNEX 20 “Air Flow Pattern within Building”, The Nederlands.
2. Loomans, M. 1992. The measurement and simulation of indoor air flow. PhD Thesis. The University of Eindhoven. The Nederlands.
3. Chen, Q and Srebric, J. 2001. Simplified diffuser boundary condition for numerical room air flow models. Final report for ASHRAE RP-1009. Department of Architecture, Massachusetts Institute of Technology. Cambridge, MA
4. Koskela, H, Heikkinen, J, Niemelä, R, Hautalampi, T. 2001. Turbulence correction for thermal comfort calculations. Building and Environment, Vol 36/2, pp. 247-255.
5. ISO 7730, 1994. Moderate thermal environments – PMV and PPD indices indices and specification of the conditions for thermal comfort. International Organisation of Standardisation. Switzerland.
6. DIN 4715-1, 1995. Chilled surfaces for room – Part 1: Measuring of the performance with free flow – Test rules. DIN. Germany.
7. ANSYS CFX, 2004. CFX Release 10 User Guide
8. Postner, J.D, Buchanan, C.R, Dunn-Rankin, D. 2003. Measurement and predictions of indoor air flow in a model room. Energy and Building, Vol 35/3 pp. 515-526.

9. Chen, Q. 1996. Prediction of room air motion by Raynolds stress models. Building and Environment, Vol 31/3, pp. 233-244.
10. Moureh, J, Flick, D. 2003. Wall air-jet characteristics and airflow pattern within slot ventilated enclosure. Interantional Journal of Thermal Sciences, Vil 42/7 pp. 703-711.
11. Stamou, A, and Katsiris, I. 2005. Verification of a CFD model for indoor airflow and heat transfer. Building and Environment Vol. 41, pp. 1171 – 1181.