# Experimental and Numerical Investigation of Air Distribution in a Large Space

Ali Alzaid*[1], Maria Kolokotroni*[1], Hazim Awbi[2]

*1 Mechanical Engineering, School of Engineering and Design, Brunel University*
*Uxbridge, Middlesex UB8 3PH, UK*
*lg15aaa4@brunel.ac.uk*
*Maria.kolokotroni@brunel.ac.uk*

*2 School of construction Management & Engineering, University of Reading*
*Reading RG6 6UA, UK*

## ABSTRACT

A literature review has revealed that there is a very limited number of numerical or experimental studies of the air flow for mechanically ventilated large occupied rooms. Existing literature suggests that a room with more than 5 meters floor-to-ceiling height can be considered as a large space. The aim of this paper is to present a set of detailed air temperature and velocity measurements in a large open plan office located in south England. External weather conditions were monitored with a meteorological station located on the roof of the building to include air temperature, relative humidity, solar radiation and wind speed/direction. The monitored office has dimensions of 15.5m x 14m x 6m ceiling height with brick external walls and metal roof which includes two large skylights. The open plan office is used by 12 people with positioned desks and computers while lighting is provided by Theroux. The total internal gain heat load in the office is 27 W/m$^2$. This large occupied open plan office is supplied by a mechanical cooling using overhead mixing ventilation system which operates during the summer months.
Measurements were carried out in the summer of 2016 which included some periods with external temperatures up to 28.9˚C and solar radiation up to 740 W/m$^2$. The measurements were used to validate a Computational Fluid Dynamic (CFD) model of the office building executed using ANSYS Fluent. A comparison between numerical results and experimental results show good agreement. The validated CFD model was used to study in more detail the existing ventilation system and proposed improvements for its performance including further commissioning.

## KEYWORDS

Large space, Air distribution, Experimental measurements, CFD model, Model validation

## 1 INTRODUCTION

Ventilation of large spaces differs from spaces with small volume, especially when the ceiling height is more than in typical spaces usually around 3m.This is because convection currents and thermal stratification have a powerful influence on the flow pattern in a large enclosure. When warm air under the effect of buoyancy escalates in a large open space, a positive temperature gradient between the floor and the ceiling will be formed which is known as stratification (Calay et al. 2000). According to Li et al. (2009), an enclosure which has more than 5 meters floor-to-ceiling height can often be considered as a large space. The air flow pattern in large enclosures should be arranged and controlled to ensure an acceptable indoor air quality in the occupied zone without the need for too much air flow rates (Heiselberg et al. 1998). Mateus & Carrilho da Graça (2017) made an extensive literature survey of existing studies of HVAC systems performance in large rooms, and they found that only three types of room air distribution strategies are used in large rooms. These are displacement ventilation, mixing ventilation and underfloor air distribution systems. Furthermore, their review divulged that there are rareness of researches which make a comparison between ventilation model simulations and measured air temperature in large spaces.

These measurements are needed for commissioning, diagnostic and assessment purposes. Unluckily, the considerable volume and envelope area associated with large enclosures adds to the difficulty of making measurements (International Energy Agency (IEA) 1998). A mixing ventilation system is a type of system where the fresh air is mixed with impure enclosure air to provide a fresh supply of air and reduce the impurity concentrations (Awbi 2011). The air jet is usually supplied in the top parts of the enclosure with velocity > 2.0 m/s to provide air circulation within the enclosure. The temperature and contaminant concentration in the room should be uniform. Whereas a displacement ventilation system is based on the fundamental of displacing the impure enclosure air with fresh outside air. The cool air usually supplied at near the floor with low velocity (normally < 0.5 m/s) to make an upward air motion (thermal plumes) as it gets warmed by heating sources in the enclosure. Therefore, vertical gradients of air velocity, temperature and contaminant concentrations will usually be created. Displacement ventilation system is usually more energy efficient compared to the mixing ventilation system because it needs lower fan power and has higher ventilation performance.

Due to the expensive and lengthy nature of experimental measurements, Computational Fluid Dynamic (CFD) is a helpful tool to conduct parametric studies. Several models can be used for simulation of turbulent flow. Reynolds-averaged Navier-Stokes (RANS) equation simulation using one and two-equation turbulence models such as RNG k-ε models, SST k-ω model and Reynolds Stress model (RSM) are often used. Furthermore, large-Eddy simulation (LES) and direct numerical simulation (DNS) are time-resolved methods which can be used for simulations. To estimate the quality of the numerical predictions a comparison between experimental results and the numerical predictions is a common procedure (Svensson et al. 2012).

The objective of this paper is to present a set of detailed air temperature and velocity measurements in a large occupied open plan office located in southern England. This large office is supplied by a mechanical overhead mixing ventilation system which operates during the summer months. The experimental data were used to validate two turbulence models RNG k-ε model and SST k-ω model. The validated model was used to study in more detail this ventilation system and proposed improvements for its performance.

## 2 DESCRIPTION OF THE CASE-STUDY AND VENTILATION SYSTEM

The large open plan office used by research staff and students was chosen as the case-study of large space because its floor-to-ceiling height is 6m.The enclosure has dimensions of 15.5m x14m x 6m and a floor area of 201 $m^2$ with brick external walls and metal roof which includes two large skylights. Two big rectangle windows are located on the south side wall of the building with dimensions 3.5m x 1.1m and 4.2m x 1.1m. There is one door on each end wall of the building. The researchers' large open plan office includes 12 personal computers, peak occupancy of 12 occupants and artificial lighting comprising of 23 luminaires each equipped with two 49 W lamps. The total internal heat gain in the office is 27 W/$m^2$. The office is supplied by a mechanical cooling overhead mixing ventilation system which operates during the summer months. The external air is delivered into the building interior through a 13m long cylindrical supply duct with 0.7m diameter. This duct has eight air diffusers located at a height of 3.7 m above the floor with dimension of 0.8m x 0.15m and divided into seven segments. Air exhaust is via two return grills located at a height of 3.7m with dimensions 1.0m x 0.5m each, see Figure 1.

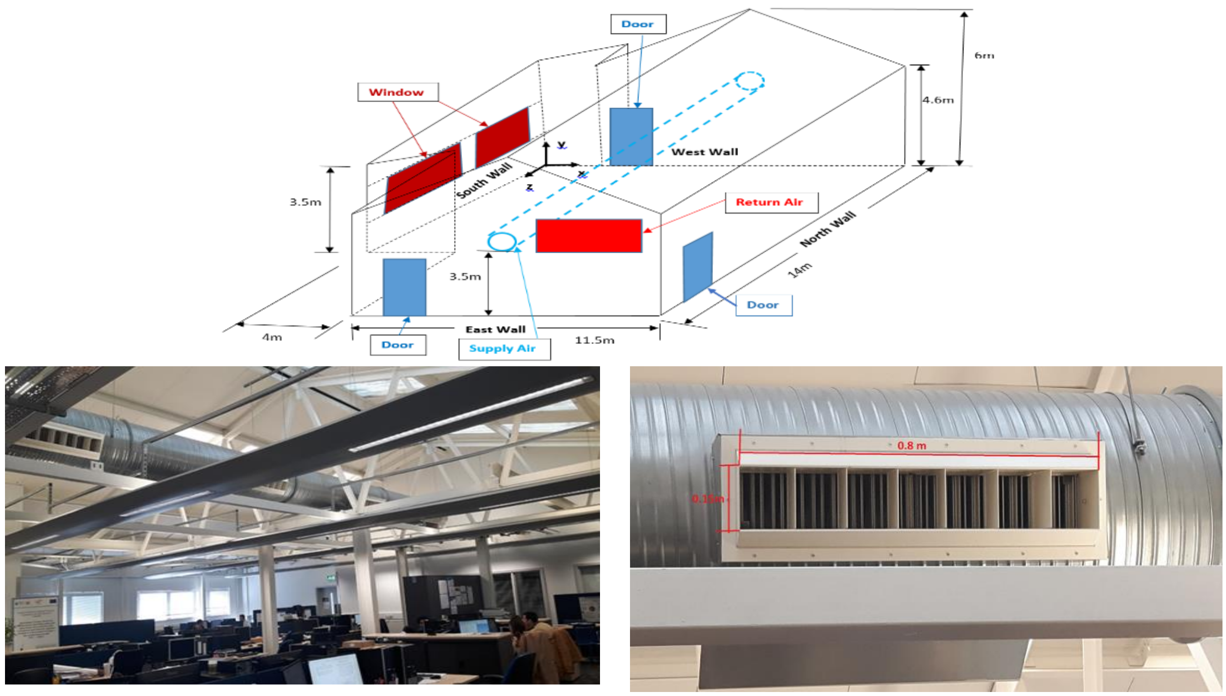


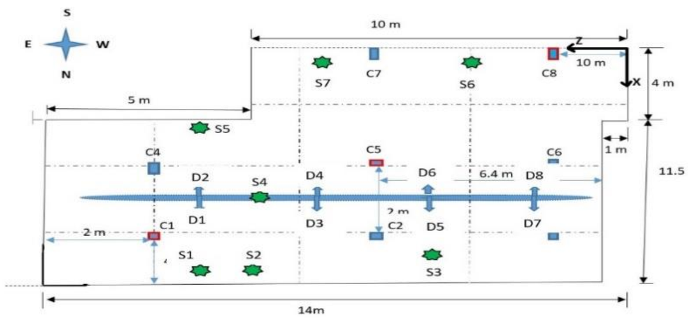
Figure 1: Sketch and photos of the researchers' office and an air diffuser at (CSEF) building

The measurements were carried out over a period of 28 days from 24/8/2016 to 21/9/2016 which can be considered late summer season in London. During this period and according to the on-site weather station the external average temperature was 18.5˚C, the maximum was 28.9˚C, the minimum temperature was 11.3˚C while solar radiation was up to 740 W/$m^2$. On the other hand, the external average relative humidity for the same periods was 80%, the maximum was 100% and the minimum was 39.4 %. In general this period of the year can be considered to be almost the hottest days in London.

## 3 EXPERIMENTAL SET-UP

Air temperature and relative humidity were measured using nine HOBO Temp/RH data loggers attached to three columns which are (C1, C5 and C8) at three different heights of 0.1, 1.2 and 1.8m, measuring temperature and relative humidity distributions between the floor and ceiling, see Figure 2. In addition, six HOBO, Temp/RH data logger, were used to measure air temperatures at the six diffusers and three more loggers were mounted at the height of 4m at three different points to measures the air temperature and relative humidity in the area above the occupied zone from 24/8/2016 to 21/9/2016. The accuracy of the air temperature measurement is ± 0.21˚C and ± 3.5% for the relative humidity measurements (HOBO n.d.).

Measurements of mean air velocity and temperature were conducted using a TA465 AirFlow instrument over five days from 5/9/2016 to 9/9/2016 at three different times of a day which were (11:00, 13:00 and 15:00). Seven different spots throughout the enclosure area have been assigned to capture vertical gradients of velocity and air temperature. These locations were chosen to describe the indoor environment in the occupied zone. The velocities were measured over two minutes with a sampling interval of ten seconds at heights of 0.1, 1.2, 1.8m at each spot. These measurements were repeated over five days three time a day. The accuracy of the velocity measurement is estimated to be ± 0.015 m/s or ± 3% while the error of measured temperature is estimated to be ± 0.3 ˚C.



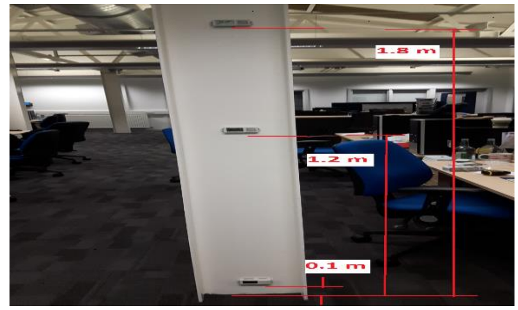



Figure 2: Schematic layout of the researchers' office at (CSEF) building and HOBO Temp/RH data logger location attached to the column at three different heights of 0.1, 1.2 and 1.8m

# 4 MEASUREMENT DATA

## 4.1 Temperature and Relative Humidity Data

Figure 3: Air temperatures for C1 at four levels external and diffuser D1 over nine days. It compares expected results for the temperatures at column C 1 for different heights 0.1, 1.2, 1.8, 4.0m as well as the temperatures at diffuser D1 plus the external temperatures. There were sharp drops in temperature in each day at 06:00 for interior temperature lines which are temperature lines at 0.1, 1.2, 1.8, 4.0m and at $D_1$ due to turning on time for the cooling system. On 03-04/09/2016, the air temperature readings were nearly the lowest compared with the other days as they were weekend days. The air temperature reached a peak on 7/9/2016 where both the diffuser and external temperature were higher than 28 ˚C.

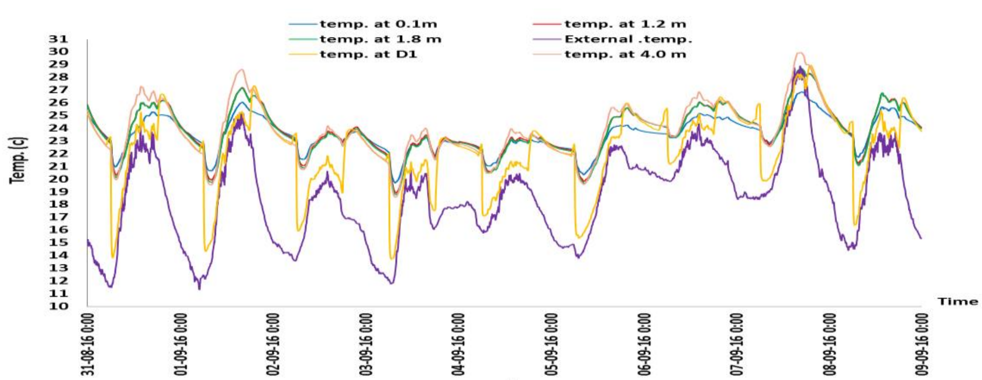



Figure 3: Air temperatures for C1 at four levels external and diffuser D1 over nine days.

There was nearly a 5˚C difference between the diffuser and outside air temperature when the cooling system was off. Whereas, the difference has been reduced to be approximately 1 ˚C when the cooling system was turned on. Also, the air temperature at the diffuser has risen sharply by 3 ˚C when the system was turned off due to hot air rising by buoyancy force. Then there was a moderate decrease in the temperature to match the temperatures at 1.2m, 1.8m and 4.0m, see Figure 4. In addition, temperatures at 1.2m were precisely the same as the temperatures at 1.8m during the ON period of the ventilation system which was from 06:00 to 18:00. However, the temperatures at 4.0m were higher than those at 0.1, 1.2m and 1.8m from 06:00 to 18:00. There was a steady state situation for all the temperatures from 15:00 to 17:00.

Figure 5 shows the temperature trends for three different locations in the enclosure which were $C_1$, $C_5$ and $C_8$ at 1.2m over 9 days as well as the external temperature. The temperature at point $C_8$ was much higher than the temperatures at $C_1$ and $C_5$ because it was very close to the external wall facing the south and this can be seen clearly at the mid-day of 7/9/2016.

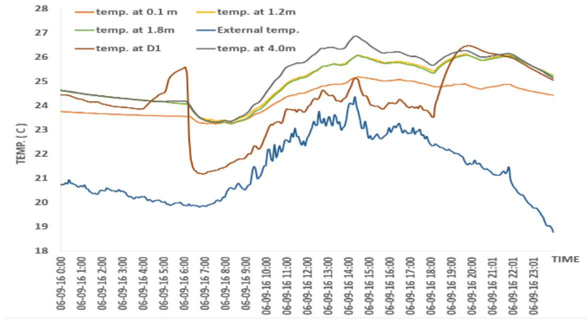



Figure 4: Air temperatures for C1 at four levels, external and diffuser D1 for one day

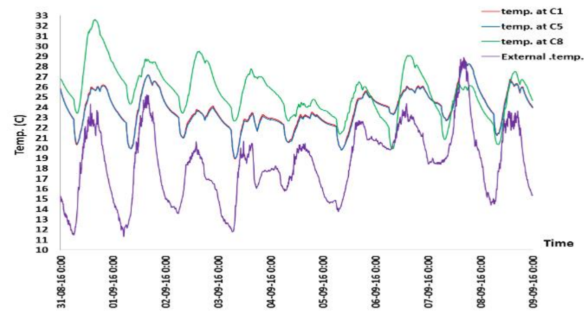



Figure 5: Temperature for C1, C5, C8 and external over nine days at height 1.2m

## 4.2 Velocity and Temperature Data

The measured velocities are presented in Figure 6 for six different spots in the enclosure. Air velocity at spots 1 and 3 show similar trend compared to the other spots because they have similar air flow direction and were located at the same distance from the diffusers. Spot 5 has the maximum velocity for both heights 0.1m and 1.2m compared to the others. Both spots 5 and 2 have similar air velocity tendency while it was different for spot 6. The velocity at spot 4 was close to 0 m/s for the three levels due to its location as it was under the main duct.

Figure 7 shows the air velocity for seven spots (see Figure 2) as measured at 1.2m height and at 15:00 over five days. The velocity at spots 1, 6 and 7 was within the acceptable range, while it was just above that range at spots 2 and 3. At spot 5, the velocities were twice the recommended velocity in the occupied zone which is 0.25 m/s.

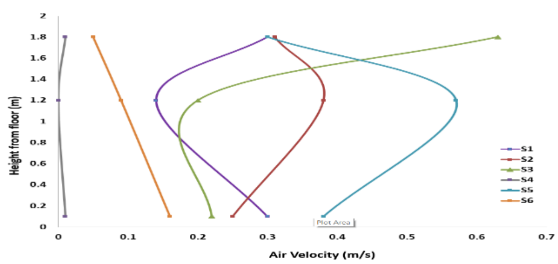



Figure 6: Air velocity in six spots at different heights

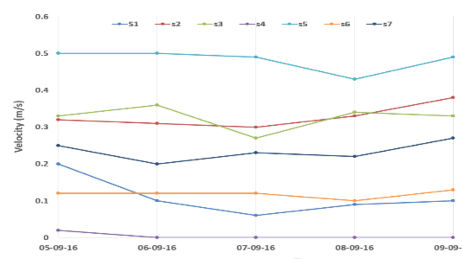



Figure 7: Air velocity in six spots at 1.2 heights over five days

Figure 8 illustrates the air temperatures for the seven spots that were mentioned above at 1.2m over five days. All the seven spots had approximately 28 ˚C as maximum temperatures on 07/09/2016 compared to the other days due to the high ambient temperature on that specific day. The air temperatures were almost the same at all the spots during the five days.

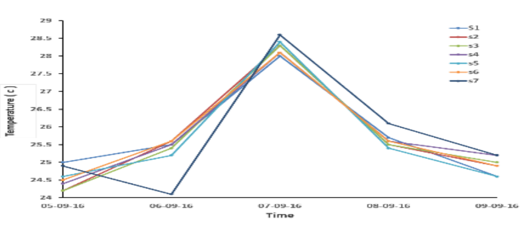



Figure 8: Air temperatures in seven spots over five days

## 5 CFD MODELLING RESULTS AND DISCUSSION

ANSYS Workbench 17.1 with Fluent 17.1(ANSYS Fluent 2016) was used to simulate the case study numerically. The fundamentals of modelling room air flow are turbulence modelling, treatment of airflow and heat convection near the wall, boundary condition, discretization of the domain and the solution method. The CFD simulation model was assumed to be steady-state and the simulation time was 15:00 h on September 6th, 2016 with four selected room locations used for further analysis as shown see Figure . These four spots are S1 which is in the north-east quarter of the office room and facing the airflow of diffuser one, S4 which is located in the middle of the office underneath the supply duct, S5 which is located in south-east quarter of the office and facing the air supply from diffuser two and S6 which is in the south-west quarter of the office and close to the windows.
Thai et al. (2007) made an evaluation of various turbulence models in predicting the airflow and turbulence in enclosed environments using CFD, and they recommended RNG k-ε model to be used in forced convection flow which is often performed in enclosures with mechanical ventilation systems. However, they recommended SST k-ω model for high buoyancy flow in predicting air velocity, temperature and turbulence quantities. For that reason, both of RNG k-ε model and SST k-ω model were used to validate the CFD model and compare their performance by utilising the temperature and air velocity measurements. Awbi (1998) pointed out that the convective heat transfer coefficient forecasted using a wall function is intensely critical to the distance of the point from the surface ($y_p$) at which the wall function is applied, i.e. the value $y^+$ at that point. He proposed in his study that an optimum position for a heated wall or heated floor is about 5 mm and about 30 mm for a heated ceiling.

### 5.1 Boundary Conditions and Physical model

The quality of the numerical solution is invariably dependent on the accuracy of the boundary conditions and how these are combined with the numerical model. In this investigation, the following boundary conditions were defined based on the measurements carried out.

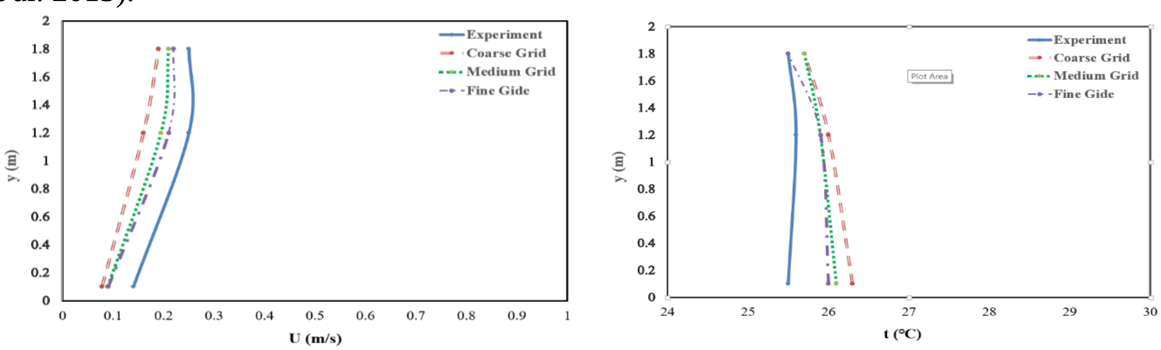
1. The condition of the air supply is set in the CFD model at values of 23$^o$C for all the eight air diffusers while the inlet velocities were not equal and modelled as seen. They were 2.72 m/s for diffuser one, 3.2 m/s for diffuser two, 2.97 m/s for diffuser three, 3.1 m/s for diffuser four, 1.0 m/s for both diffusers five and six and 0.95 for both diffusers seven and eight based on measured data. These values were chosen to correspond with the measured values.
2. The wall temperatures were measured at different points and were found to be between 28 and 29 $^o$C. Whereas the ceiling and window temperatures were 35 and 39 $^o$C respectively.
3. The body of the occupant was presented as a cylinder of height 1.4m and diameter of 0.4 m giving a body surface area of approximately 1.85 $m^2$ according to (Pinkel 1958). The clothing temperature is set to a value of 33.7 $^o$C as mentioned by Zolfaghari & Maerefat (2010).

The ANSYS code was used to construct the three-dimensional geometry and generate the mesh. Non-uniform grid strategy was utilised to cover the whole computational domain for the room. The finer grid was used close to inlet, outlet and walls, and also the areas that were anticipated to have the steep velocity gradient. A grid independency study was performed using the RNG k-ε model, and three grids densities, i.e. (15, 082, 503), (16, 045, 809) and (16, 791, 869) were tested. The results are discussed in the following section.

The finite-volume solver Fluent 17.1 was used to simulate the flow field of the ventilated enclosure. The governing equations were solved with a segregated scheme, and the SIMPLE algorithm solved the coupling between pressure and velocity. The discretization, the non-linear and the viscous terms were calculated with second order upwind scheme while the BODY FORCE WEIGHTED scheme was used to reveal convergence when two consecutive iterations for the local variable was less than $10^{-3}$ whereas for energy it was less than $10^{-6}$. Besides that, the net mass flow rate imbalance was less than 0.003% of the total flux through the system, and the net heat imbalance was less than 0.3% of the total energy flux through the system too.

## 5.2 Grid Independency Study

The mean air velocity and temperature profiles along the height of 0.1, 1.2, 1.8m of the spot (S2) for three mesh densities are shown in Figure 9 (a) and (b). It can be seen that the predicted mean temperature gradient for of the three meshes are incredibly fused with less than 0.4% difference between the two finer meshes, which indicates the coarse mesh is still fine enough at the three heights for temperature prediction. In addition, the predicted mean velocity along three heights at spot (S2) are compared for the three tested meshes, and finer two meshes show small difference which is about 3%.Thus, the medium grid has been chosen based on their two comparisons (Chen et al. 2013).




**(a)** **(b)**

Figure 9: Comparisons between simulation results profiles along the height of 0.1, 1.2, 1.8m of the spot (S2) for three mesh densities (a) mean velocity and (b) mean temperature.

## 5.3 CFD Validation with Measurement

Figure 10(a) and (b) shows the comparisons of the mean air velocity and temperature profiles between simulation results using the two turbulent models and experimental data at four difference spots. Figure 10(a) shows both turbulence models are able to predict the velocity distributing satisfactory when compared with the experimental results. Nonetheless, the less satisfactory agreement with measurement was observed in the spot (S5) at the height 1.2m. This is due to its location which is near to the one of the room doors. The predicted temperature profile is indicated in Figure 10(b) in which model SST k-ω shows slightly better agreement with measurement than the RNG k-ε model. The high-temperature discrepancy between the prediction and measurement was up to 3.5℃ in the spot (S6) for RNG k-ε model probably due to its location near the sunny windows. Extra heat flux may be conveyed into the office and help to heat up the air around a spot (S6). To sum up, the prediction from SST k-ω show better agreement with measurements in comparison with RNG k-ε model. Hence, all results presented in the next section are based on SST k-ω turbulence model.

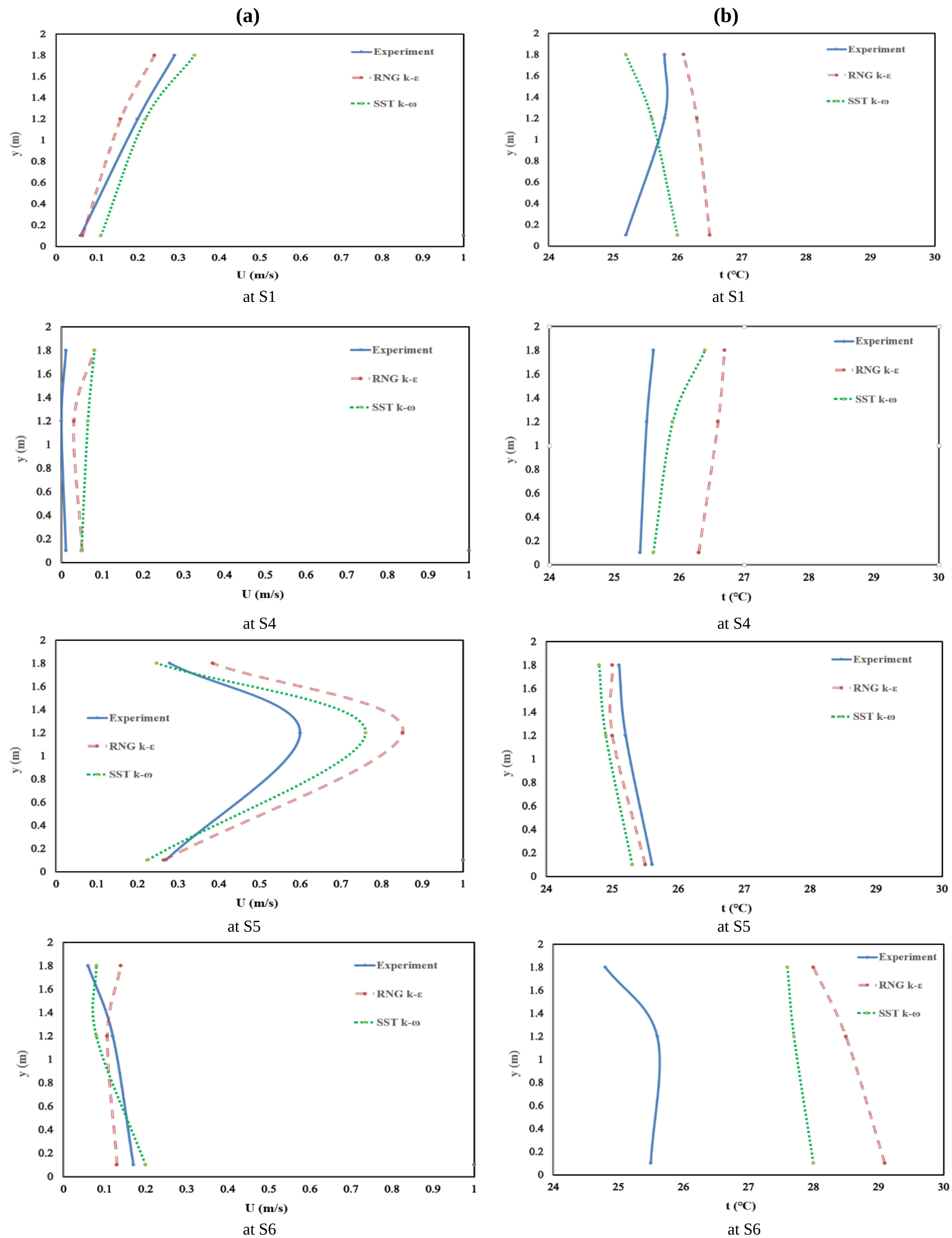
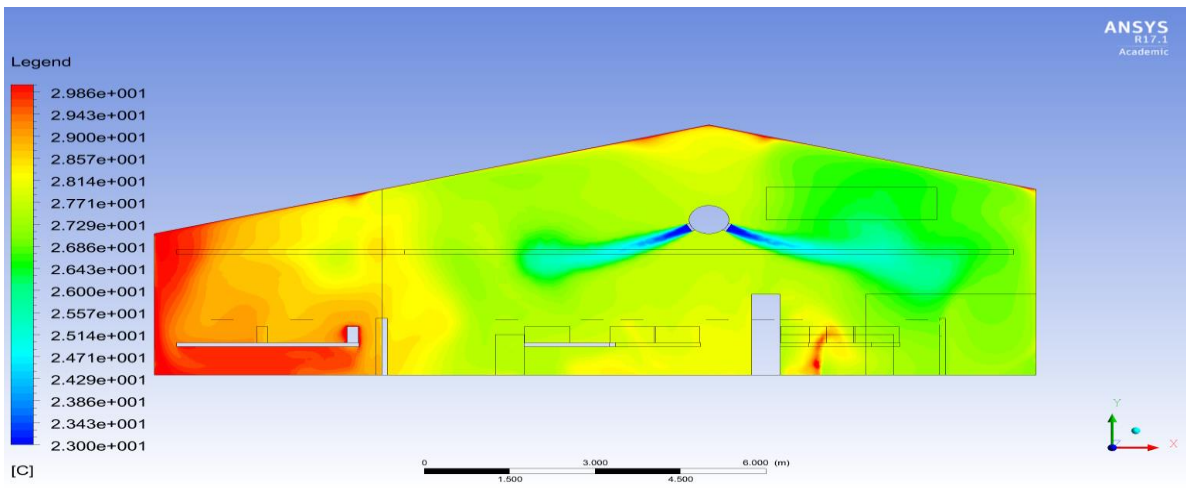



Figure 10: Comparisons between simulation results with measurements along three heights for a spot (S1, S4, S5 and S6) (see Figure 2): (a) mean velocity and (b) mean temperature.

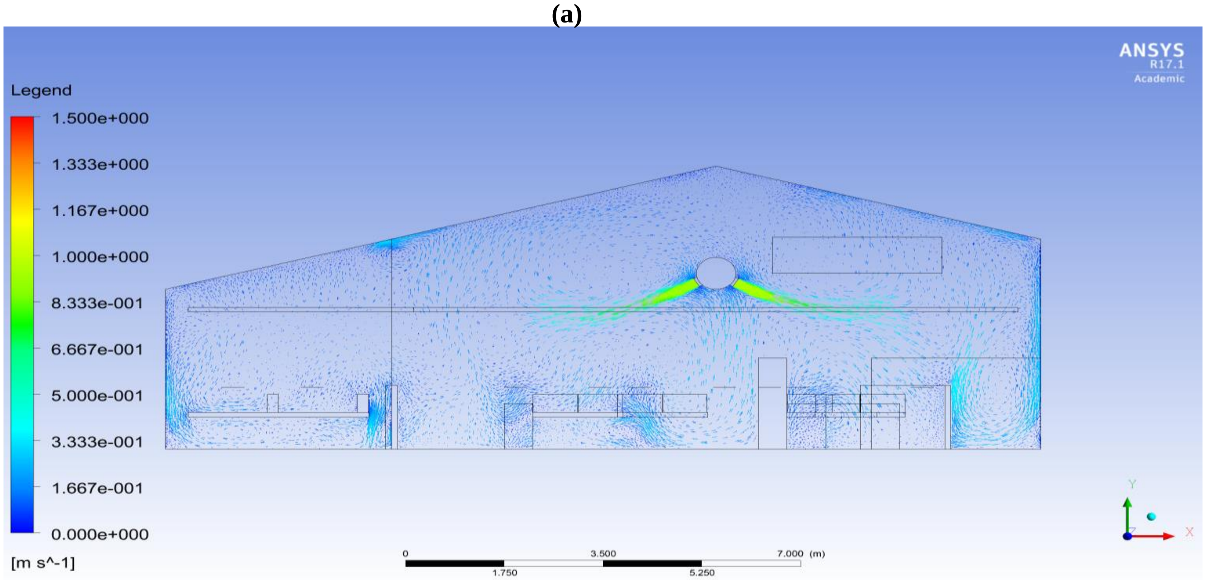
## 5.4 Results from Case Study

The qualitative numerical results indicating the temperature contour for a cross-section (at 4m) from the west wall in Figure 2 is shown in Figure 11(a), where an apparent temperature stratification is observed. The cold air supply gets warmer while it spreads along the office. It can be observed that the temperature tends to rise continuously along to the south wall. A convective plume is created when the air approaches the solar heated wall. It should be noted that the mean temperature at height 4m was nearly the same as the measured mean temperature (26°C).

Figure 11(b) displays the predicted velocity vector fields for the same cross-section. A cross-recirculation is created in different parts of the occupied zone of the office due to the partitions and furniture. The window thermal plume generates strong buoyancy force which moves the air up to the ceiling. On the other side of the room, the main flow is merged with the downward generated flows from the supply duct which increases the momentum of the air penetrating into the office.




**(a)**




**(b)**

Figure 11: Cross-section (4m) from the west wall in (Figure 2): (a) Temperature counter (b) Vector velocity.

## 6 CONCLUSION

In this paper, full-scale measurements of a mechanically ventilated large space have been carried out. First, the validation of two turbulence models RNG k-ε and SST k-ω was performed. According to the results, both models were capable of somewhat capturing the main flow features when results were compared the experimental findings. However, the predictions from SST k-ω model were slightly better for the mean velocity values. The validated CFD model was therefore used to study the room ventilation system. In order to modify the current system, it was found that the eight air diffusers should have the same air velocity to achieve optimum air distribution. On the whole, the current study is a step towards understanding and later improve currently used ventilation systems for large spaces by studying different types of ventilation systems such as the impinging jet system.

## 7 REFERENCES


ANSYS Fluent, 2016. Thory guide.

Awbi, H.B., 1998. Calculation of convective heat transfer coefficients of room surfaces for natural convection. *Energy and Buildings*, 28(2), pp.219–227.

Awbi, H.B., 2011. Energy Efficient Ventilation for Retrofit Buildings retrofit degli edifici. , pp.23–46.

Calay, R.K., Borresen, B.A. & Holdø, A.E., 2000. Selective ventilation in large enclosures. *Energy and Buildings*, pp.281–289.

Chen, H.J., Moshfegh, B. & Cehlin, M., 2013. Investigation on the flow and thermal behavior of impinging jet ventilation systems in an office with different heat loads. *Building and Environment*, 59, pp.127–144.

Heiselberg, P., Murakami, S. & Roulet, C., 1998. Ventilation of large spaces in buildings. *Final Report IEA Annex*.

HOBO, Onset HOBO and InTemp Data Loggers. Available at: http://www.onsetcomp.com/ [Accessed May 16, 2017].

International Energy Agency (IEA), 1998. IEA Annex 26: Energy Efficient Ventilation of Large Enclosures - Technical Synthesis Report.

Li, Q. et al., 2009. CFD study of the thermal environment in an air-conditioned train station building. *Building and Environment*, 44(7), pp.1452–1465.

Mateus, N.M. & Carrilho da Graça, G., 2017. Simulated and measured performance of displacement ventilation systems in large rooms. *Building and Environment*, 114, pp.470–482. Available at: http://dx.doi.org/10.1016/j.buildenv.2017.01.002.

Pinkel, D., 1958. The use of body surface area as a criterion of drug dosage in cancer chemotherapy. *Cancer research*, 18(3), pp.853–856.

Svensson, K. et al., 2012. Numerical and Experimental Investigation of Flow Behavior in a Confluent Jet Ventilation System for Industrial Premises. *DiVA Portal*, pp.1–6.

Thai, Z.Q. et al., 2007. Evaluation of various turbulence models in predicting airflow and turbulence in enclosed environments by CFD: part 1 - Summary of prevalent turbulence models. *Hvac&R Research*, 13(6), pp.853–870.

Zolfaghari, A. & Maerefat, M., 2010. A new simplified thermoregulatory bioheat model for evaluating thermal response of the human body to transient environments. *Building and Environment*, 45(10), pp.2068–2076.