Simulation of Airflow in Lecture Rooms

Johnson Lim Soon Chong[1], Adnan Husain[2] & Tee Boon Tuan[3]
[1]Faculty of Technical Education, [2]Faculty of Mechanical Engineering & Manufacturing
Kolej Universiti Teknologi Tun Hussein Onn, Batu Pahat, Johore, Malaysia.
[3]Faculty of Mechanical Engineering,
Kolej Universiti Teknikal Kebangsaan Malaysia, Ayer Keroh, Malacca, Malaysia.

ABSTRACT

The purpose of this study is to perform simulation of airflow in lecture rooms using Computational Fluid Dynamics (CFD) technique. Two lecture rooms with different location of air-conditioning units are chosen as study fields for measurements and simulation purposes. The air distribution parameters addressed in this study are air velocity and air temperature. Measurements were also done on the boundary parameters of the room such as air-conditioning inlet and outlet conditions. The simulations were performed by using CFX software without involving occupants. Two types of turbulence models were compared for suitability in lecture room modeling. Validation of results through linear regression technique was performed using SPSS software. Based on the stream plots, the zones below the air-conditioning units were identified to have inadequate airflow. In overall, the RNG $k-e$ Turbulence Model yields better prediction than Reynolds Stress Turbulence Model. Simulation can be a useful technique to predict air distribution performance in a lecture room for a better thermal environment.

Keywords: Airflow, Simulation, Modeling

(1) INTRODUCTION

Most of the students spend majority of their time in lecture rooms during lecture or tutorial. Air flow distribution from the air-conditioning system becomes important in this case, as it will determine whether students perceive the air velocity and air temperature in the acceptable comfort range.

This study emerged based on a need to further investigate the current airflow distribution system in lecture rooms of KUiTTHO. The airflow in a typical lecture room in KUiTTHO is not uniform for the current arrangement of air-conditioning units in the room. It was also discovered that the position of air-conditioning units in the lecture rooms will affect the air flow distribution in the room, which then causes changes in thermal sensation among the occupants [1].

There are some locations in the lecture room where occupants feel warm; and some other locations where occupants feel cold. This is mainly due to the improper selection of air-conditioning unit location during the installation of the air conditioning system. Therefore, a CFD study is necessary in order to investigate the airflow pattern and airflow distribution, which will help to identify the locations where air distribution is inadequate.

Based on literature studies, there is less study of room airflow modeling which involves lecture rooms. Furthermore, majority of the CFD room modeling involves modeling of rooms such as cold storage room, telecommunication room and operating room where only little emphasis is given to total air distribution performance and occupant's thermal sensation. Other studies involved rooms where important room parameter can be controlled [2-3]. However, lecture rooms (where KUiTTHO is concerned) are mostly occupied by students and their perceived thermal sensation are dependable on the thermal environment created by the air-conditioning system in the room. Therefore, the study of the airflow modeling is crucial towards achieving desirable thermal environment in the lecture room.

The main interest of this research study is to simulate airflow in the lecture rooms of KUiTTHO, either with or without occupants. The main objectives are as follows:

(a) To identify and obtain important modeling parameters for valid CFD room modeling.
(b) To evaluate air distribution performance in lecture rooms through measurements and CFD techniques.

(c) To compare and validate results obtained from both the measurements and CFD simulation techniques.

(2) METHODOLOGY

The scope of this study includes pilot and actual measurements of the important air distribution parameters and the simulation performed by using a CFD software. The actual physical conditions in the room are maintained without alterations during measurements. The main procedures of this study are as follows:

(a) Selection of suitable lecture rooms
(b) Performing actual measurement on important air distribution parameters and boundary parameters
(c) Performing CFD analysis on room air flow through computer simulation

Before performing any physical measurements in a typical lecture room, the lecture room is cleaned. Eight rows of long chairs are arranged in both sides in the room, according to a fixed arrangement, as defined by the writer. The whole room is arranged accordingly. A typical fixed arrangement for a lecture room is shown in **Fig.1**. The exact location of each rows of long chairs, lecturer's table, air-conditioning unit and measurement points. All the other obstacles in the lecture room including overhead projector (OHP), OHP screen and extra long chairs are taken out of the room during measurement. Curtains were closed in an attempt to eliminate heat transfer through glass window, as shown in **Fig.2**. All measurements were done without the presence of occupants.

Two lecture rooms with different location of air-conditioning units are chosen as study fields for measurements and simulation purposes. In the case of Lecture Room 1 (BK 17) and Lecture Room 2 (BK 23), the air-conditioning units were installed at the side of the rooms. Two rooms, each with different supply location, will then be chosen for actual simulation purpose. The dimensions for the mentioned lecture rooms are standard with 9.64 m long, 7.11 m wide and 3 m height.

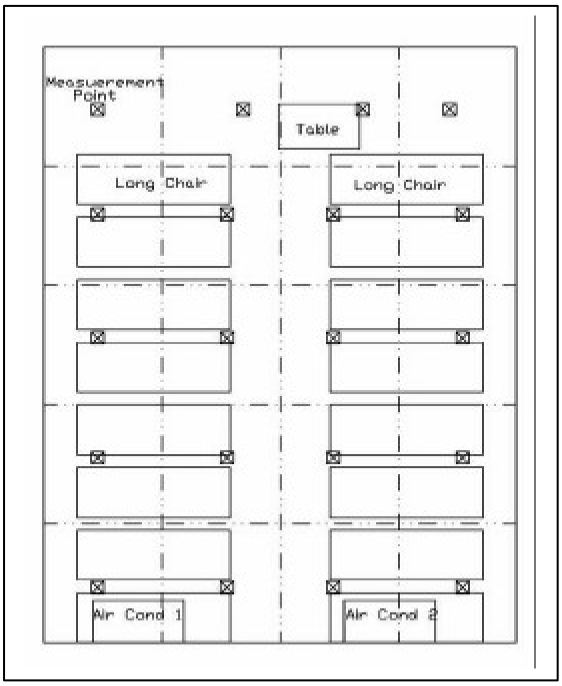

Fig.1 *A Typical Fixed Arrangement for Lecture Room.*

Fig.2 *Closed Curtains to Minimize Heat Transfer through Windows.*

In order to assess the lecture room's air distribution performance using ADPI, the lecture room has been divided into 20 zones, following the methods used by Tee [1]. The zones are dotted lines that divide the room into 20 zones, as shown in **Fig.1**. The points of measurements are located at the center of each zone, on a fixed coordinate of location.

Before performing measurement using anemometer, a typical lecture room is let to stabilize for one hour, with reference to the methodology used by Tee [1]. The purpose is to make sure that the air-conditioning system had stabilized in order to produce stable airflow for measurements. After the room is stable in one hour, the anemometer was let to stabilize for 5 minutes in the room for environment adaptability before taking the real data at each measurement point.

The time and height of measurements are following the standard as described in ASHRAE Standard 55-92 [4]. The probe of the anemometer was set at the height of 1.1 meters for both pilot measurement and actual measurement. As the point of measurements is near long chair, the anemometer's probe is fixed at the long chair using adhesive tape. For other areas, the anemometer is fixed at a normal plastic chair. For a typical point of measurements, the time to record readings is 3 minutes. The data taken for a particular point of measurements are air velocity and air temperature.

Simulation Procedures

The first step in CFD modeling is the model building. The whole lecture room was drawn geometrically following real physical measurement data, as described in Appendix A. In addition, the lecturer's table, long chairs and two types of air-conditioning units were also modeled according to their own actual physical measurements in CFX. **Fig. 3** shows a lecture room with side location of air-conditioning units modeled in CFX. Lecture room with back location of air-conditioning units as modeled in CFX is also shown in **Fig. 4** All the simulations were also done without the presence of occupants.

The second step in modeling is to define the fluid domains. A fluid domain is the area where fluid flow is defined. In CFX, The fluid used was set to "Air at STP". The reference pressure was 101325 Pa, the atmospheric pressure. The simulation type was steady state with stationary domain motion. For the fluid models, either the RNG κ - ε Turbulent Model or Reynolds Stress Turbulent Model were chosen. For each room, these two models were used to identify a better turbulent model for room air flow prediction.

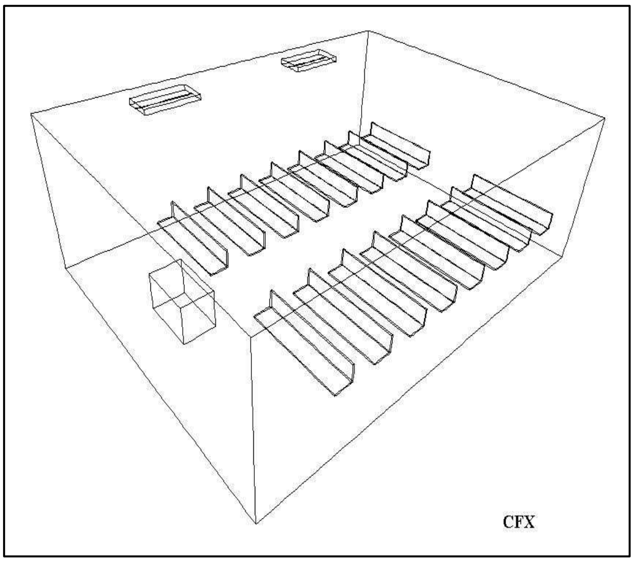

Fig.3 *Lecture Room with Side Location of Air-Conditioning units in CFX.*

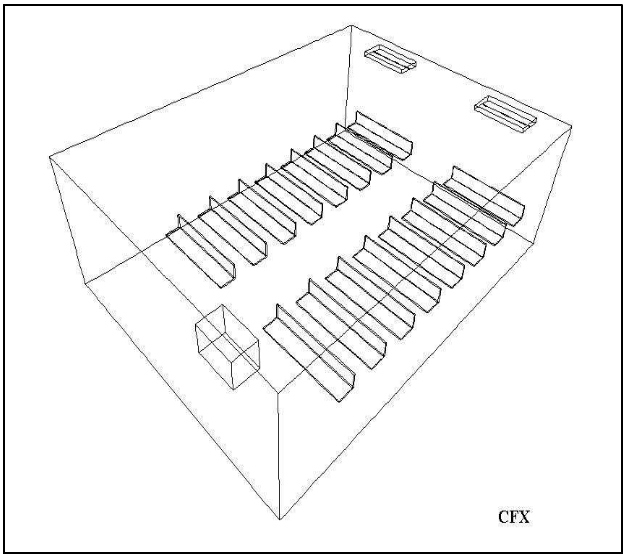

Fig.4 *Lecture Room with Back Location of Air-Conditioning units in CFX.*

Data Analysis Method

Upon collecting all the actual measurement data, the data was analyzed using a few methods. As each room was divided into 20 zones with 20 points of measurement, the Effective Draft Temperature (EDT) was evaluated for each point using the air velocity and air temperature obtained. ADPI was then evaluated for each room. Upon choosing the two lecture rooms to be modeled in CFX, the two rooms were modeled in CFX to obtained predicted values of air velocity and air temperature. Based on these data, predicted EDT was obtained and predicted ADPI was then evaluated for each room.

The actual results were compared with the simulated results by using overall average error calculations and absolute error. The equations are:

$$E_{ABS} = \frac{\left| X^{i}_{CFD} - X^{i}_{\exp} \right|}{X^{i}_{\exp}} \times 100\% \qquad \ldots\ldots(1)$$

$$E = \frac{\sum_{i=1}^{n} \left| X^{i}_{CFD} - X^{i}_{\exp} \right|}{\sum_{i=1}^{n} X^{i}_{\exp}} \times 100\% \qquad \ldots\ldots(2)$$

Where X can be either air velocity or air temperature. | X^{i}_{CFD} - X^{i}_{exp} | is the absolute difference between simulated values and actual measurement values for variable X. Linear regression analysis was also carried out to validate the results. Both the Microsoft Excel 97 Spreadsheet Program and the SPSS Statistical Package, Release 10.0 were used for linear regression analysis. From the analysis, the governing equation that links between the actual and the simulated values can be determined.

The residual squared value, R^2 and the Pearson's Correlation Coefficient, R can be determined and serve as a factor between the degree of linear relationship between the two variables studied. The R values ranges from –1 to 1, where –1 indicates strong negative linear relationship and 1 indicates strong positive linear relationship. If R is 0.8, it means that 80% of the data were explained by linear relationship. In order to interpret the various values of R, Guilford's suggested interpretations of R-values could be used, as shown in **Table 1**.

Table 1 *Guilford's Suggested Interpretations for Values of R [6].*

R values	Interpretation
Less than 0.20	Slight; almost negligible relationship
0.20 to 0.40	Low correlation; definite but small relationship
0.40 to 0.70	Moderate correlation; substantial relationship
0.70 to 0.90	High correlation; marked relationship
0.90 to 1.00	Very high correlation; very dependable relationship

During the course of this study, there are a few assumptions being made. These assumptions are:

(a) The air-conditioning systems studied are all in good and stable operations.
(b) The measurements and simulation are done in a completely confined room space, with all the doors and windows closed.
(c) There is no construction leakage or noticeable gap, which leads to outdoor air infiltration.
(d) The effects of physical condition of the room (floor construction, difference in location of glass windows and doors, etc.) are neglected in this study.
(e) The anemometer used is well calibrated and in good conditions during measurement.
(f) The airflow produced in the room studied is steady during measurements, where steady state conditions are assumed for simulation.
(g) The heat transfer through glass windows and doors are neglected. Only the heat transfer through wall and ceiling are considered.
(h) Internal heat sources in the lecture room such as fluorescent lights are considered to have small effects and thus neglected in study.
(i) The outside temperature is constant at 30 °C during measurements.
(j) Air velocity of 0.10 m/s is considered as still air and air velocity difference of below than 0.10 m/s as insensible [5] and small.
(k) ADPI of 60 to 69 is considered as unsatisfactory, 70 to 79 as satisfactory and 80 and above as good air distribution.

(3) RESULTS AND DISCUSSION

A stream plot displays streamlines, which can be thought of as the paths of imaginary mass-less particles through the geometry in a steady-state calculation. This is usually useful for showing the particle paths through the geometry. In this study, since the RNG $\kappa - \varepsilon$ Turbulence Model is a better model for lecture room model representation, the stream plots for lecture room BK 17 and BK 23 in isometric view are obtained and shown in **Fig. 5** and **Fig. 6**.

From the stream plots for lecture room BK 17 in **Fig. 5**, it can be observed that the cold air drops in the front location of the room for air-conditioner no. 1 and air-conditioner no. 2. Upon dropping of air, mixing of air occurs in the front location of the lecture room as a result of cold air recirculation. The middle of the room experienced good mixing of air. The back location of the room, particularly the area below the air conditioning units, experienced poor mixing of air where air flow is inadequate. This resulted in a stagnant area at the back of the room, where relatively higher temperature and lower air velocity values at location 17, 18, 19 and 20.

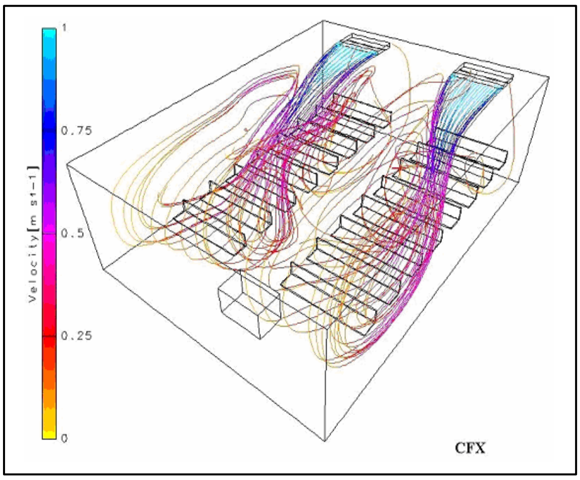

Fig. 5 *Stream Plots for RNG k – e Turbulence Model Simulation in Lecture Room BK17..*

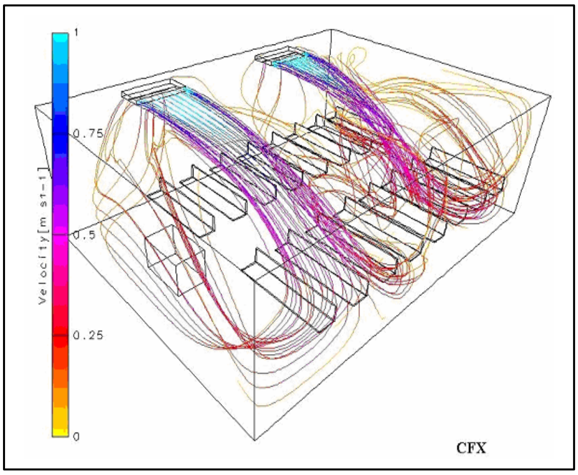

Fig. 6 *Stream Plots for RNG k – e Turbulence Model Simulation in Lecture Room BK 23.*

The validity of results can be judged in terms of the absolute difference, overall average error percentage and the results from linear regression analysis. In terms of difference of in air velocity, it was assumed that the difference in air velocity of below 0.10 m/s as insensible and small. This assumption was made following the study done by Kubo *et.al* [5]. This study also assumed that the temperature difference of below 1°C is small.

Table 2 shows in summary the percentage of results with absolute difference of air velocity is below 0.10 m/s. It was found that RNG $\kappa - \varepsilon$ Turbulence Model produced 85% of air velocity results with difference of below 0.10 m/s, in both simulations in lecture room BK 17 and BK 23. The Reynolds Stress Turbulence Model had 70% and 80% of the results with difference below 0.10 m/s in simulations of BK 17 and BK 23. In this case, the RNG $\kappa - \varepsilon$ Turbulence Model performs slightly well than Reynolds Stress Turbulence Model.

Table 2 *Percentage of Results with Absolute Difference of Air Velocity Value of Below 0.10 m/s.*

Lecture Room	Turbulence Model	Percentage of Results < 0.10 m/s (%)
BK 17	RNG $\kappa - \varepsilon$	85
	Reynolds Stress	70
BK 23	RNG $\kappa - \varepsilon$	85
	Reynolds Stress	80

Table 3 *Percentage of Results with Absolute Difference of Air Temperature Value of Below 1 °C.*

Lecture Room	Turbulence Model	Percentage of Results < 1 °C (%)
BK 17	RNG $\kappa - \varepsilon$	100
	Reynolds Stress	45
BK 23	RNG $\kappa - \varepsilon$	90
	Reynolds Stress	65

For air temperature difference, it is shown in **Table 3** that for simulation in BK 17 using RNG $\kappa - \varepsilon$ Turbulence Model, all the difference of simulation results with the actual values are below 1 °C, which is small in value. For simulation in BK 23, 90% of the difference in simulation results are below 1 °C. Comparing with the simulations using Reynolds Stress Turbulence Model, there are only 45% and 65% of the simulation results with difference below 1 °C. Therefore, it can be concluded that RNG $\kappa - \varepsilon$ Turbulence Model predicts air temperature more accurately than Reynolds Stress Turbulence Model.

The comparison between overall average error percentage in this study with the previous study conducted by Hoang *et.al* [3] are shown in **Table 4** and **Table 5**. Only air velocity results were compared as it was the only relevant results with study conducted by Hoang *et.al* [3]. The comparison revealed that the overall average error percentage for this study is higher than overall average error percentage by Hoang *et.al* [3].

Table 4 *Comparison of Overall Average Error Percentage for Air Velocity with the Previous Study.*

	Present Study				Hoang *et.al* (1999)
Simulation	RNG κ – ε, in BK 17	RNG κ – ε, in BK 23	Reynolds Stress, in BK 17	Reynolds Stress, in BK 23	RNG κ – ε, in Cold Storage Room
Overall Average Error Percentage (%)	34.87	32.81	72.62	53.31	28.5

Table 5 *Comparison of Overall Average Error Percentage for Air Temperature in Present Study.*

	Present Study			
Simulation	RNG κ – ε, in BK 17	RNG κ – ε, in BK 23	Reynolds Stress, in BK 17	Reynolds Stress, in BK 23
Overall Average Error Percentage (%)	1.49	2.58	4.74	3.92

With regards to air distribution performance of lecture rooms, both lecture room BK 17 and lecture room BK 23 has the same measured ADPI value of 70, which is satisfactory in terms of air distribution performance. The RNG κ – ε simulation predicted lower ADPI value in lecture room BK 17 (ADPI of 65), but higher ADPI value in lecture room BK 23 (ADPI of 75).

(4) CONCLUSIONS

The simulation of airflow in lecture rooms was successfully carried out. Measurement results were used to compare with the simulation values. This study showed that the RNG κ – ε Turbulence Model predicts better than Reynolds Stress Turbulence Model and therefore is suitable as turbulence model-to-model lecture room. Majorities of over 80% of the simulation values are found to have small difference with actual measurement values in terms of air velocity and air temperature. Overall average error percentage of simulations done in this study is all over 30%, which is higher than the overall average error percentage produced by previous study.

Nevertheless, linear regression analysis showed good agreement between simulation values and measurement values. This is evident from the values of Pearson's correlation coefficient, R of between 0.7 and 0.9 for each simulation results produced with RNG κ – ε Turbulence Model. It is therefore concluded that high correlation and marked relationship exist between simulation results and measurement results.

This showed that the boundary conditions in the model are defined correctly. The important boundary conditions that should be defined for correct modeling are the inlet-outlet conditions of the air-conditioning units and the heat transfer properties by the ceiling and wall of the room model.

The stream plots produced by CFD simulation can serve as a useful tool to observe airflows in room. It can also show regions with inadequate airflow that leads to poor mixing of room air. Based on the stream plots, the area under the air-conditioning units showed inadequate air flow, which causes lower air velocity and higher air temperature. In general, simulation can be a useful technique to predict air distribution performance in a lecture room for a better thermal environment.

(5) REFERENCES

[1] B.T. Tee, *Effects of Airflow Towards Thermal Comfort in Lecture Rooms,* B.Eng. Thesis, Universiti Teknologi Malaysia, 2001.

[2] P. Depecker, *et. al*., " Study And Comparison Of Two Types Of Air Flow In Operating Rooms Using A CFD Code", *Proceedings of the 7th Int. Conference On Indoor Air Quality and Climate: Indoor Air 96*, July 1996, Nagoya Japan, Vol. 2, pp. 719 – 723,1996.

[3] M.L. Hoang, *et. al*., " Analysis of the Air Flow in A Cold Store By Means Of Computational Fluid Dynamics", *International Journal of Refrigeration*, Vol. 23, pp. 127 – 140, 1999.

[4] ASHRAE." ASHRAE Standard Thermal Environment Conditions for Human Occupancy." Atlanta: (55-1992R). 2001

[5] H. Kubo, *et.al.*, " Human Response To The Different Body Surface Area Exposed To Airflow", *Proceedings of the 7th Int. Conference On Indoor Air Quality and Climate: Indoor Air 96*, July 1996, Nagoya Japan, Vol. 2, pp. 553 – 558, 1996.

[6] R.C. Sprithall, *Basic Statistical Analysis(4th Edition)*, Massachusetts, Allyn and Bacon, 1993.

[7] T.H. Kuehn, *et.al*., *Thermal Environment Engineering*, New Jersey, Prentice-Hall, 1998.

[8] M. Loomans, *The Measurement and Simulation of Indoor Air Flow*, Ph.D. Thesis, Eindhoven University of Technology,1998.

[9] M. Ramey, *Application of Computational Fluid Dynamics to Indoor Room Airflow*, M.Sc. Thesis, Oklahoma Christian University of Science and Arts, 1994.