Indoor Airflow Simulation inside Lecture Room: A CFD Approach

S Lin, B T Tee\textsuperscript{1} and C F Tan
Centre for Advanced Research on Energy (CARe)
Faculty of Mechanical Engineering, Universiti Teknikal Malaysia Melaka,
Hang Tuah Jaya, 76100 Durian Tunggal, Melaka, Malaysia.

E-mail: tee@utem.edu.my

Abstract. Indoor air flow distribution is important as it will affect the productivity of the occupants. Poor air flow distribution not only cause discomfort to the occupants but also influence their ability to conduct their activities. The main purpose of this study is to investigate the indoor air flow inside lecture rooms through CFD simulation approach. Two types of air-conditioning configuration system in lecture rooms have been selected for this study which includes the split unit and centralized system. The air flow distribution between these two systems are analyzed and compared. Physical measurement is conducted using a velocity meter for validation purpose. CFD simulation is developed by using ANSYS Fluent software. The results specifically the air velocity and temperature data are compared and validated. Based on the findings, design recommendation is proposed with the aim to improve on the current air flow distribution in the lecture rooms.

1. Introduction

Most of the students spend majority of their time in lecture rooms during their lecture or tutorial session. In this case, air flow distribution from the air-conditioning system becomes important as it will determine whether students perceive the air velocity and air temperature in the acceptable comfort range [1]. This study emerged based on a need to further investigate the current air flow distribution system in lecture rooms of UTeM. The air flow in a typical lecture room in UTeM 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. Therefore, a Computational Fluid Dynamics (CFD) study is necessary in order to investigate the air flow pattern and air flow distribution, which will help to identify the locations where air distribution is inadequate. This has been supported by previous studies (Lim et.al [1], Lim et.al [2], Cheong et.al[3], Ruponen & Thinker [4] and Gon et.al [5]).
2. Methodology

2.1 Room Selection Procedure
Two lecture rooms (Figure 1 & Figure 2) with different type of air conditioning configuration have been selected for this study. The details of the rooms are given in Table 1 below:

| Items                    | Lecture Room 1 (LR1) | Lecture Room 2 (LR2) |
|--------------------------|----------------------|----------------------|
| Dimension (m)            | 12 (L) x 9 (W) x 3 (H) | 14.85 (L) x 7.35 (W) x 3.5 (H) |
| Air-conditioning Configuration | Central Unit     | Split Unit          |

![Figure 1. Lecture Room 1 (LR1)](image1)

![Figure 2. Lecture Room 2 (LR2)](image2)

2.2 Indoor Air Parameter Measurement
The indoor air parameter which to be measured in this study are the air velocity and air temperature. The air velocity meter was used for data measurement and the height of the sensor was adjusted to 1.1 m above ground level. The range of air temperature for this device is from 17.8°C to 93.0°C while the range of air velocity is from 0 m/s to 30m/s. The selected rooms were divided into 20 zones to obtain the desired measurements as suggested by previous studies [1] [2].

![Figure 3. Zonal division of LR1 (left) and LR2 (right)](image3)
2.3 Simulation Procedures
The simulation is conducted by using the ANSYS Fluent CFD software. From this study, it is learned that the general room modeling procedures including defining geometry of the model, defining fluid domains, boundary conditions, initial guess, meshing and solver control. The assumptions made for simulation are given as follows:
(a) The air conditioning system in selected lecture rooms is fully functioning and running well.
(b) The rooms are fully sealed and enclosed without any holes or gaps (excluding doors, windows and exhaust vent).
(c) The outside temperature on the surface of the room is constant
(d) Internal heat source emitted from the digital devices and lights will be neglected due to minimal effect on the temperature.
(e) The furniture (chairs and tables) are included in the simulation.

The simulation parameter is summarized in Table.

| Items                      | LR1                        | LR2                        |
|----------------------------|----------------------------|----------------------------|
| Domain                     |                            |                            |
| Viscous model              | k-epsilon (2 equation)     | k-epsilon (2 equation)     |
| k-ε Model                  | RNG                        | RNG                        |
| Near-wall treatment        | Standard Wall Functions    | Standard Wall Functions    |
| Meshing                    |                            |                            |
| Number of Nodes            | 30779                      | 27886                      |
| Number of Elements         | 155986                     | 141285                     |
| Solution Method            |                            |                            |
| Turbulent Kinetic Energy   | Second Order Upwind        | Second Order Upwind        |
| Turbulent Dissipation Rate  | Second Order Upwind        | Second Order Upwind        |

Figure 4. Geometry sketch of LR1 (left) and LR2 (right).
3. Results and Discussions

Based on the experimental results as shown in Table 3, the average air temperature for LR1 was 27.3 °C and the average temperature for LR2 was 21.8 °C. The average air velocity for LR1 was 0.12 m/s and the average air velocity for LR2 was 0.15 m/s. Referring to this data, the results for LR1 does not conform to the standard MS 1525:2007 [6] and further consideration can be taken to improve the overall air-conditioning system. The results for LR2 however are reasonably good with better air flow in the lecture room.

According to CFD approach, the analysis was successfully performed to obtain the air parameters for both lecture rooms. The simulation modeling was influence with the standard procedures which are defining the geometry of the lecture room, meshing, fluid properties, boundary conditions, solver settings, solution methods and the results are obtained from CFD post. The average air temperature for LR1 was 26.9°C and the average temperature for LR2 was 18.7°C. The average air velocity for LR1 was 0.10 m/s and the average air velocity for LR2 was 0.18 m/s. Similar with the results from the physical measurement, the data obtained for LR1 and LR2 does not conform to the standard of MS 1525:2007.

Figure 5 and 6 shows the CFD stream plot of the air velocity inside both rooms. Based on the figures, it can be seen that the air distribution pattern for centralized air conditioning unit performed fairly better than the split unit system even though the air velocity present in LR1 was lower.

| Table 3. Summary of results of Lecture Room 1 and Lecture Room 2 |
|---------------------------------------------------------------|
| **Items** | **Recommended Standard MS 1525:2007** | **Physical Measurement of LR1** | **CFD Simulation of LR1** | **Physical Measurement of LR2** | **CFD Simulation of LR2** |
|-------|---------------------------------|----------------------------|----------------|----------------------------|----------------|
| Average Air Velocity (m/s) | 0.15-0.5 | 0.12 | 0.10 | 0.15 | 0.18 |
| Average Temperature (°C) | 22-26 | 27.3 | 26.9 | 21.8 | 18.7 |

*Figure 5. Stream plot for LR1.*
The orientation and positioning of the split unit air conditioning unit in LR2 could be altered to obtain a better air flow pattern. Figure 7 below shows that a different orientation of the air conditioning unit could give a better air flow pattern in the lecture room.

4. Conclusions
The comparison results for LR1 between physical measurement and CFD simulation gives 16.7% difference for air velocity and 1.4% for the temperature. The results for LR2 give percentage difference of 20.0% for air velocity and 14.22% for temperature. The errors that exist are mainly due to the uncontrolled infiltration in actual room which difficult to be quantified for CFD simulation. In term of air-conditioning configuration, the air flow distribution pattern in LR1 has a better streamline pattern than LR2. Even though the air velocity in LR1 is below than the recommended standard, the air distribution tends to be steady and well distributed across every part of the room.
Acknowledgment

The authors would like to thank Universiti Teknikal Malaysia Melaka for the financial support through Short Term Research Grant award (PJ/2012/FKM(6A) S01077.

References
[1] Lim J, Tee B T & Husain A 2005 Simulation of Airflow in Lecture Room The AESEAP International Conference 2005 (Kuala Lumpur, Malaysia 7th – 8th June 2005) (Kuala Lumpur, Malaysia:Universiti Malaya)
[2] Lim C M, Tee B T, & Lim J 2006 Airflow Simulation Studies Inside KUTKM's Lecture Rooms National Seminar on Science and Its Applications in Industry (Malacca, Malaysia 14th – 15th February 2006) (Malacca:KUTKM) p 27 – 39.
[3] Cheong K, Djunaedy E, Tham C K, Sekhar S, Wong N & Ullah M 2003 Building and Environment 38(1) 63–73
[4] Ruponen M & Tinker J A 2007 CFD Validation of the thermal comfort in a room using draft rates. Proceedings of the Clima 2007 Wellbeing Indoor Congress (Helsinki, Finland 10th - 14th, 2007) (Helsinki:Finland) ISBN 978-952-998-3-6-6
[5] Kim G, Schaefer L, Lim T S & Kim J T 2013 Energy and Buildings 64 323–331
[6] Malaysian Standard 2007 Code and Practice on Energy Efficiency and Use of Renewable Energy for Non-Residential Buildings (Putrajaya:Department of Standards Malaysia)