A Study on Possibility of CFD Simulation on Air Simulation in Minor Operation Theatre

M N Rahman Y*, Z M Razlan, M Izham, M I Omar, N A A Zambri, Shahriman A B, I Zunaidi and W K Wan

School of Mechatronic Engineering, Universiti Malaysia Perlis, Pauh Putra Campus, 02600 Arau, Perlis, Malaysia.

*Corresponding author: nurrahman@unimap.edu.my

Abstract. Inadequate air flow distribution in room is undesired as it will cause discomfort to the occupant. Fair air flow distribution is important as it will affect the performance of the occupant as well. The objective of this research work is to develop a realistic Computational Fluid Dynamics (CFD) model of Minor Operation Theatre for investigating the air flow pattern in Minor Operation Theatre. One unit of Air Conditioner in Minor Operation Theatre has been selected for this study. The distribution of air flow pattern is analysed. The real measurement is conducted using a digital air flow anemometer with USB interface and the channels digital handheld temperature thermometer meter data loggers for validation purpose. CFD simulation is conducted by applying ANSYS Fluent software. The result of measurement and simulation are compared and validated. Based on the simulation and experiment findings, it proofs that ANSYS Fluent can be used for air simulation and relocation recommendation is proposed with the purpose for better distribution of air flow in Minor Operation Theatre.

1. Introduction

A research work for better Minor Operation Theatre (OT) concurrently is running as development for better quality of air. The surgeons and patients are expecting comfortable environmental of operation room as they spend most of their time in that room [1][2][3][4]. This research work execution based on a need to develop a realistic CFD model of Minor Operation Theatre room and pursue by investigating the pattern air flow distribution in the Minor Operation Theatre of UniMAP [2][5]. The pattern of air flow in the Minor Operation Theatre in UniMAP was assumed is not uniform for current location of AC unit in the Minor Operation Theatre. It was also recognized that the location of AC unit in the Minor Operation Theatre will affect the distribution of air flow in the Minor Operation Theatre and then influences the change of thermal sensation among occupants. Therefore, an identification on the locations where air distribution is inadequate needs a Computational Fluid Dynamics (CFD) method as a tool to investigate the air flow pattern and distribution as well. This has been supported by previous research works [6][7][8][9].

2. Methodology

2.1. Minor Operation Theatre Data Collection

One of Minor Operation Theatre at Universiti Malaysia Perlis Health Centre (Figure 1) with Spilt Unit of Air Conditoning has been selected for this study. The details of the room are given in Table 1 as below:

Table 1. The details of the room.

Items	Minor Operation Theatre
Dimension (m)	2.89 (H) x 3.21(W) x 3.4(L)
Air-conditioning Configuration	Single Unit

Figure 1. Minor Operation Theatre

2.2. Collection Data of Air Characters in Minor Operation Theatre

The velocity and temperature of air were chosen as the air characters that need to be measured. The digital air flow anemometer with USB interface was applied for air flow velocity data collection and that device were directly located at AC unit nozzle part. Meanwhile, the channels digital handheld temperature thermometer meter data loggers were applied for the air temperature data collection. The Minor Operation Theatre was divided into 9 zones vertically and each zone was divided into 4 points horizontally as shown Figure 2. The height of each sensors was adjusted to 0.5 m above from the floor level.
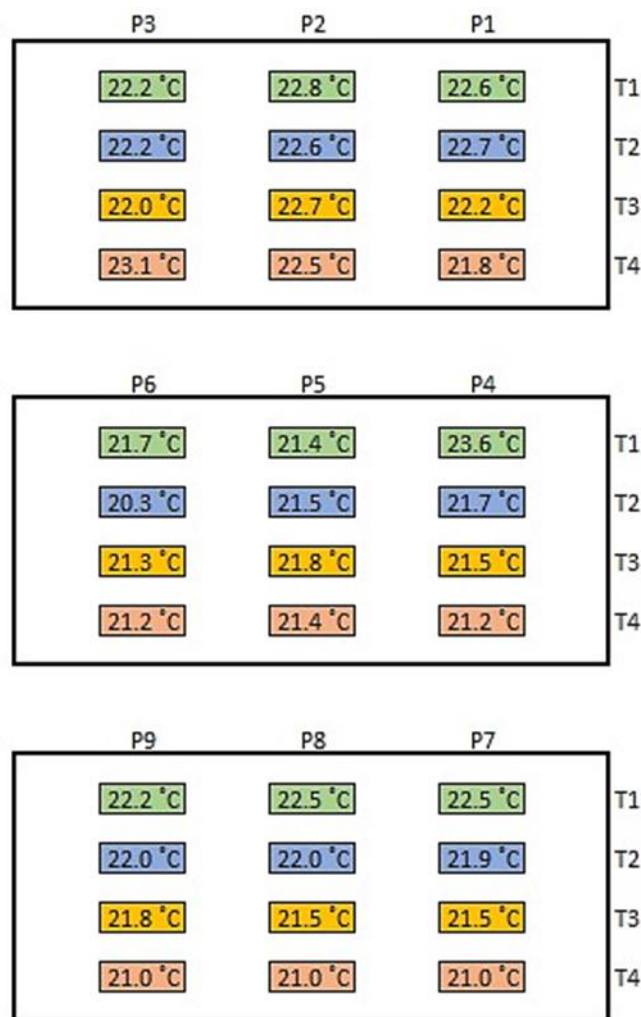
Figure 2. Cross section of the room with temperature distribution

2.3. Computational Fluid Dynamics Simulation Methodology

The commercial software named ANSYS Fluent 17.1 was utilized for simulation work in this study. In this study, the Minor Operation Theatre room (Figure 3) modeling must go through Pre-Processing, FLUENT-Solver and finally FLUENT Post-Processing for Minor Operation Theatre as shown in figure 4. The assumptions must be done while doing this simulation work due to commercial software limit and to reduce calculation time. The assumptions for this study are given as follows:

- The AC unit in Minor Operation Theatre rooms is well maintained, fully functioning and running well.
- The measurements and simulation are done in a completely sealed OT space, with all the doors and windows closed.
- The outdoor temperature on the surface of the room is constant
- Internal heat source emitted from the desktop, digital devices and lamps will be neglected due to minimal effect on the temperature.
- The effects of physical condition of the room (type of floor construction, difference in location of glass windows and doors) are neglected in this study.
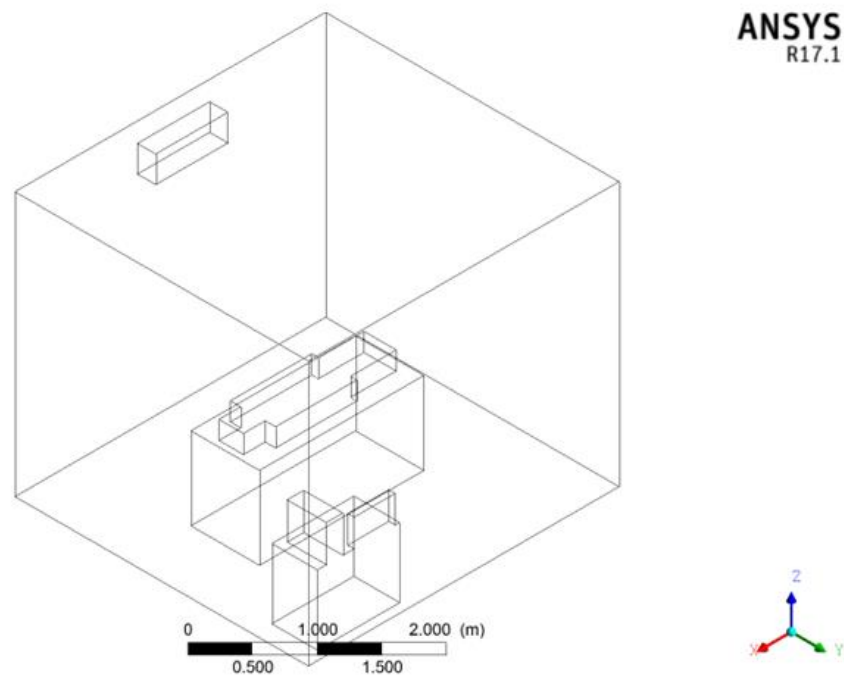- A chair, table, and bed are included in this simulation.

Figure 3. Geometry Model of Minor Operation Theatre.

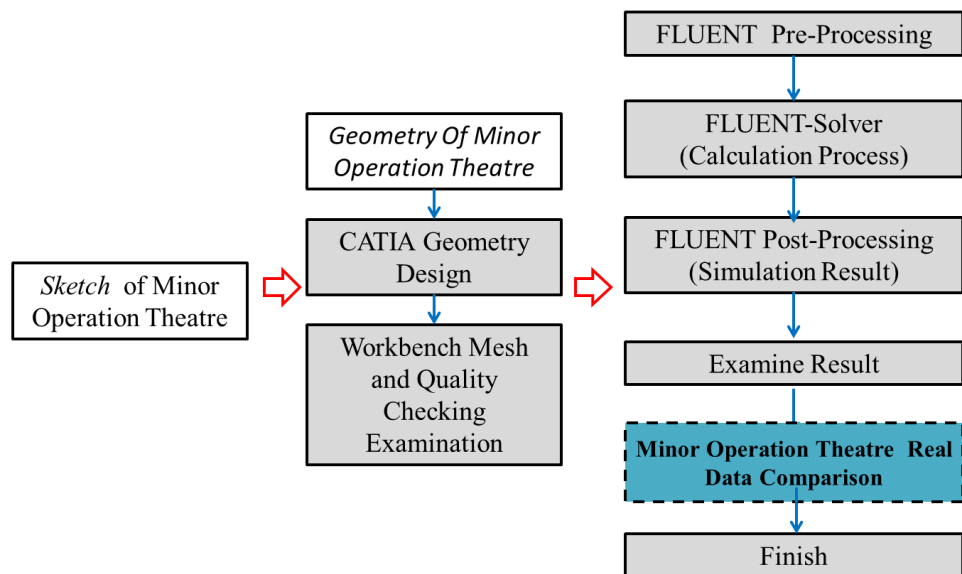**Minor Operation Theatre Simulation Flow**

Figure 4. Flow chart of Minor Operation Theatre Simulation Work.

The simulation details are summarized in table 2.

Table 2. Summary of simulation work details.

Details	Minor Operation Theatre	
General	Gravity Force	On
	Energy	On
Models	Viscous model	K-Epsilon (2 Equation)
	k-Epsilon Model	RNG
	Near-Wall Treatment	Standard Wall Functions
Boundary Conditions	AC Nozzle Velocity	2.8 m/s
Mesh Size	Number of Nodes	96469
	Number of Cells	112933
	Gradient	Least Squares Cell Based
	Pressure	Second Order
Solution Method	Momentum	Second Order Upwind
	Turbulent Kinetic Energy	Second Order Upwind
	Turbulent Dissipation Rate	Second Order Upwind
	Energy	Second Order Upwind

3. Result and Discussions

Based on the real measurement result as shown in table 3, the average airflow temperature for Minor Operation Theatre was 21.0 ºC and the average air velocity for Minor Operation Theatre was 2.52 m/s. Meanwhile the average airflow temperature and the average air velocity for CFD Simulation of Minor Operation Theatre are 17.0 ºC and 2.62 m/s respectively. The CFD approach was successfully performed to obtain the velocity and temperature of air flow in Minor Operation Theatre.

The simulation results were related to the standard methods which are redefining the geometry of the Minor Operation Theatre room, optimization in mesh density, boundary conditions setting, solver setting, right solution methods and acceptable results were obtained from CFD post.

Figure 5 and 6 shows the CFD stream plot of the air velocity inside Minor Operation Theatre and the slices of temperature contour of Minor Operation Theatre. Based on the both figures (Figure 5 and 6), single unit AC shows inadequate distribution in air flow pattern in Minor Operation Theatre cause inadequate distribution in cold air.

Table 3. Summary of results of Minor Operation Theatre

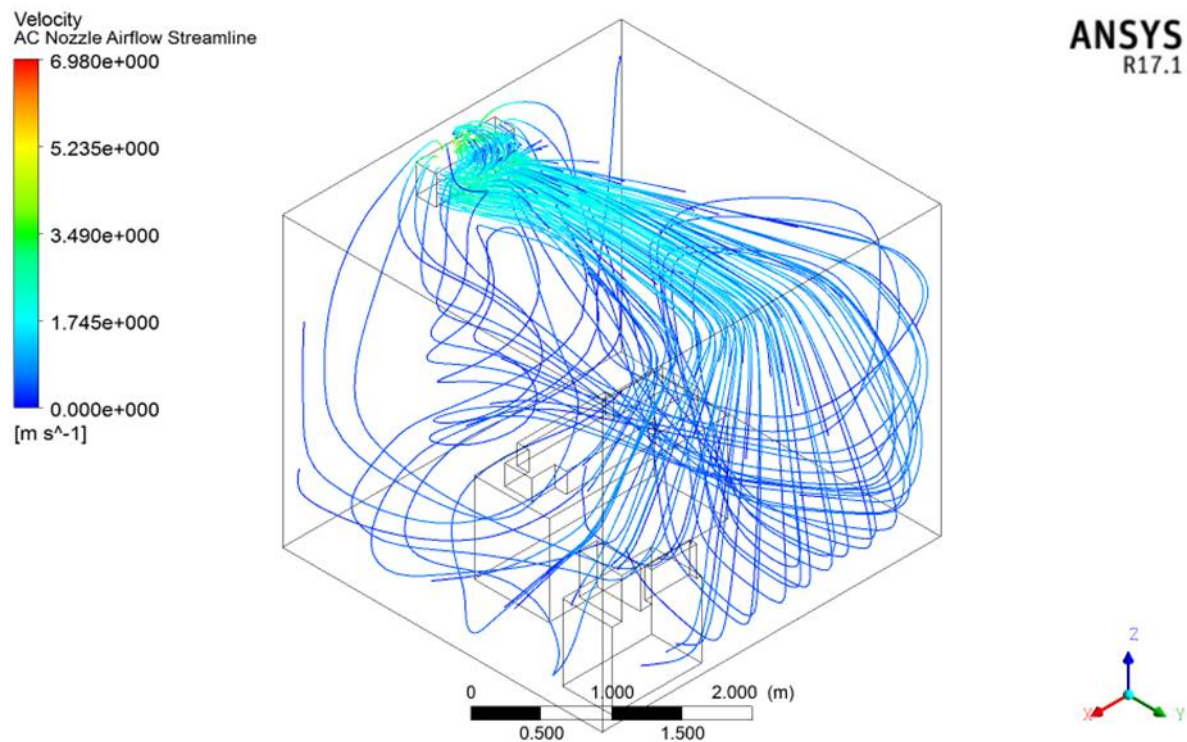| Details	Real Measurement of Minor Operation Theatre	CFD Simulation of Minor Operation Theatre
Average Air Temperature (°C)	21.0	17.0
Average Air Velocity (m/s)	2.52	2.62

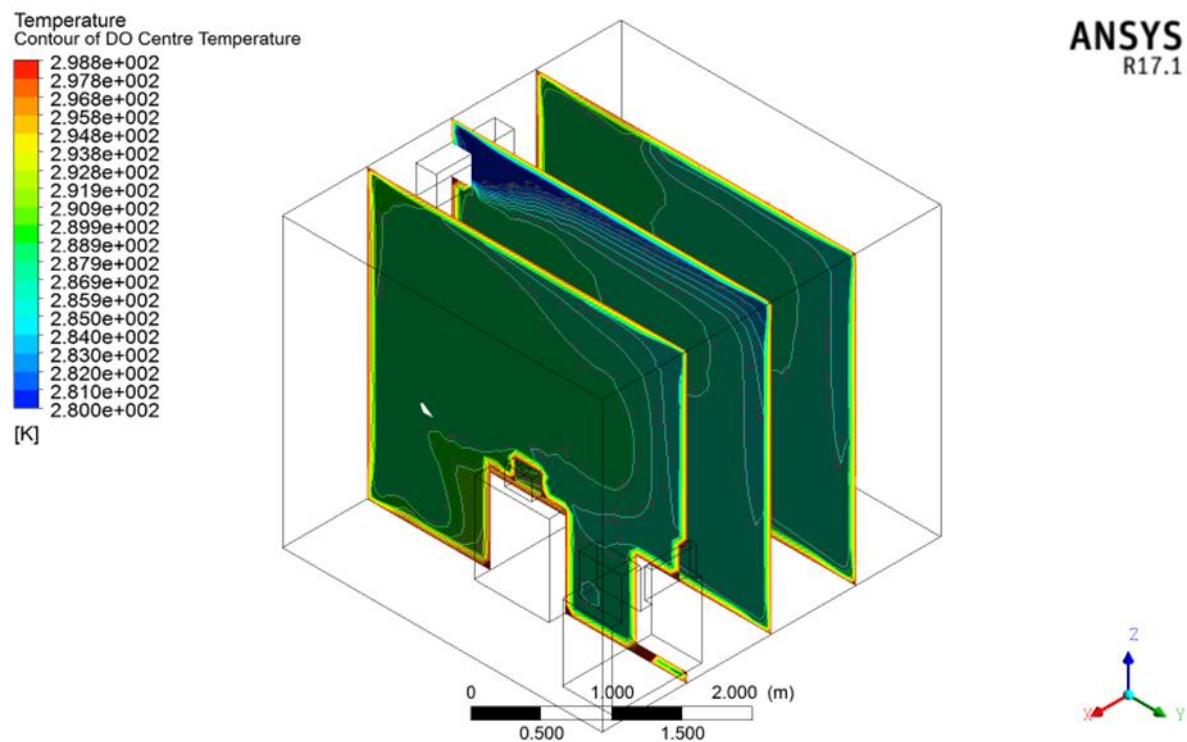**Figure 5.** AC Nozzle Airflow Streamline for Minor Operation Theatre.

Figure 6. Slices of Temperature Contour of Minor Operation Theatre based on location in figure 2.

4. Conclusion

The CFD simulation result for Minor Operation Theatre demonstrated differences in real measurement about 19% for air flow velocity and for the 0.04% temperature. The errors that occurred due to the uncontrolled mesh density and wall of Minor Operation Theatre in actual Minor Operation Theatre which hard to be quantified for CFD simulation. Based on the observation, better location of AC unit must be proposed for better distribution of air flow in Minor Operation Theatre and it reveals that ANSYS Fluent can be utilized for air simulation in Minor Operation Theatre.

References

[1]   R. Khatri, A. P. Singh, and V. R. Khare, "Identification of Ideal Air Temperature Distribution Using Different Location for Air Conditioner in a Room Integrated with EATHE - A CFD Based Approach," *Energy Procedia*, vol. 109, no. November 2016, pp. 11–17, 2017.

[2]   A. M. Paul Danca, Florin Bode, Ilinca Nastase, "On the possibility of CFD modeling of the indoor environment in a vehicle," in *Sustainable Solutions for Energy and Environment, EENVIRO 2016, 26-28 October 2016, Bucharest, Romania*, 2017, pp. 656–663.

[3]   S. Sadrizadeh, *Design of Hospital Operating Room Ventilation using Computational Fluid Dynamics (PhD thesis)*, no. February. 2016.

[4]   S.A. Nada, H.M. El-Batsh, H.F. Elattar,N.M. Ali, "CFD investigation of airflow pattern, temperature distribution and thermal comfort of UFAD system for theater buildings applications," *J. Builiding Eng.*, vol. 6, pp. 274–300, 2016.

[5]   T. C. Stevenson, "Experimental investigation of hospital operating room air distribution (MSc Thesis)," no. May, p. 97, 2008.

[6]   S. Lin, B. T. Tee, and C. F. Tan, "Indoor airflow simulation inside lecture room: A CFD approach," *IOP Conf. Ser. Mater. Sci. Eng.*, vol. 88, no. 1, pp. 1–7, 2015.

[7]   A. Abduladheem, K. S. M. Sahari, H. Hasini, W. Ahmed, and R. A. Mahdi, "Ventilation Air Distribution in Hospital Operating," vol. 2, no. 11, pp. 81–85, 2013.

[8]   Yuanda Cheng, Jianlei Niu, Naiping Gao, "Stratified air distribution systems in a large lecture theater : A numerical method to optimize thermal comfort and maximize energy saving," *Energy Build.*, vol. 55, pp. 515–525, 2012.

[9]   J. Lim, S. Chong, A. Husain, and T. B. Tuan, "Simulation of Airflow in Lecture Rooms."