

A Test of Possibility on Relative Humidity Function in Minor Operation Theatre

M N Rahman Y*, Z M Razlan, M Izham, M I Omar, N A A Zambri, Shahrizan 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. High humidity air flow in Minor Operation Theatre (MOT) is undesired as it will cause discomfort to the occupant. Acceptable humidity air flow is important as it will affect the pureness of the indoor environment as well. The objective of this research work is to construct a realistic Computational Fluid Dynamics (CFD) model of MOT for investigation. The MOT which is occupied with one unit of Air Conditioner has been selected for this study. The RH of air is analysed. The experimental data is collected by applying 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 experimental result and simulation is compared and validated. Based on the simulation and experiment findings, it reveals that ANSYS Fluent can be used for RH in MOT.

Keywords. Computational Fluid Dynamics Simulation, Minor Operation Theater, Relative Humidity.

1. Introduction

A refinement research for better air quality in Minor Operation Theatre (MOT) currently is being conducted by many researchers. The ideal and comfortable environment in operation theater always being expected by the surgeons and patients as they spend most of their time in that room [1][2][3][4]. This research is an extension study from previous work which is to construct a realistic CFD model of MOT by investigating the Relative Humidity (RH) of air flow in the MOT of Universiti Malaysia Perlis (UniMAP)[2][5]. The RH of air flow in the MOT was numerically simulated and experimentally measured. It was also assumed that the lack of Interior Design (ID) of MOT will affect the RH of air flow and then influences the required range of RH which is between 20% and 60%. Therefore, an identification on achievement of RH in MOT needs a Computational Fluid Dynamics (CFD) method and then will be verified with experimental data. This has been supported by previous research works [6][7][8][9].

2. Methodology

2.1. CFD Model of MOT

Model of MOT has been constructed by CATIA (Figure 1) and numerically simulated by ANSYS Fluent. This commercial software has capability to predict the incompressible, compressible, laminar and turbulent fluid flow along with buoyancy and compressibility phenomena. Given Table 1 as the details of the room.



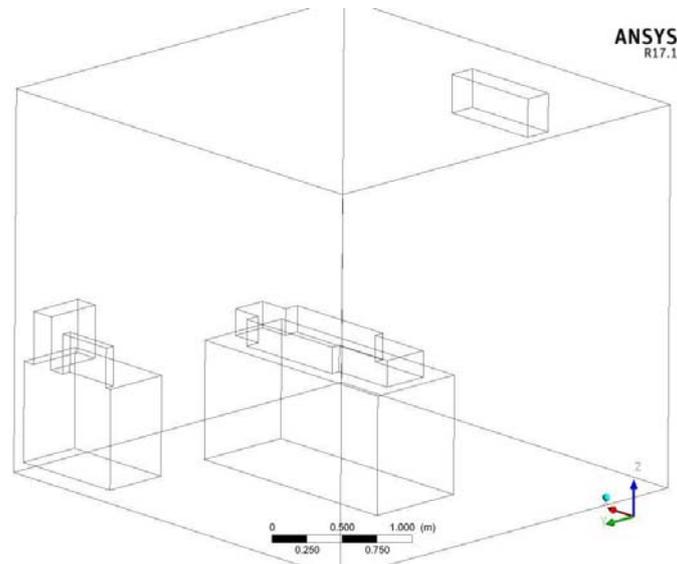


Figure 1. MOT Model (CATIA)

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

2.2. CFD 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 1) modelling must go through Pre-Processing, FLUENT-Solver and finally FLUENT Post-Processing for Minor Operation Theatre as shown in figure 2. 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 MOT is steady condition.
- MOT space is completely sealed.
- The outdoor temperature on the surface of the room is constant
- Internal heat source emitted from the electronic devices and electric appliances will be neglected due to minimal effect on the temperature.
- The effects of physical condition of the room are neglected in this study.
- The furniture is included in this simulation.

The simulation details and meshed model are summarized in table 2 and shown in figure 3.

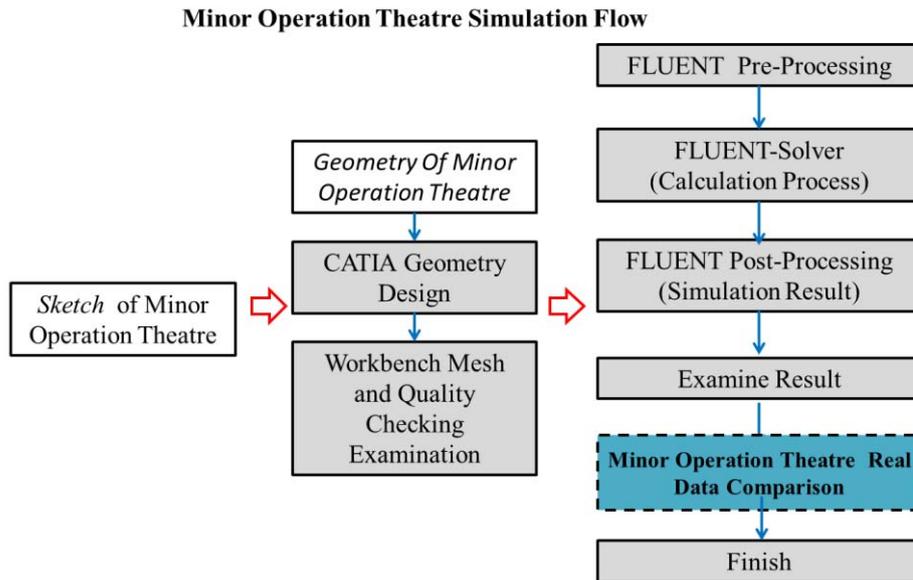


Figure 2. Flow chart of MOT Simulation Work.

Table 2. Summary of simulation work details.

Details	Minor Operation Theatre	
General	Gravity Force	On
MOT Models	Physical Model	3D
	Energy	On
Fluid Models	Viscous model	K-Epsilon (2 Equation)
	k-Epsilon Model	RNG
	Near-Wall Treatment	Standard Wall Functions
	AC Nozzle Velocity	2.8 m/s
Boundary Conditions	Air Temperature	6.9°C
	Mass Fraction of Air	99.2%
	Mass Fraction of H ₂ O	0.8%
	Relative Humidity	47%
Mesh Size	Type of Cells	Hexahedral
	Number of Nodes	167024
	Number of Cells	213814
Solution Method	Gradient	Least Squares Cell Based
	Pressure	Second Order
	Momentum	Second Order Upwind
	Turbulent Kinetic Energy	Second Order Upwind
	Turbulent Dissipation Rate	Second Order Upwind
	Energy	Second Order Upwind



Figure 3. Meshed Model.

2.3. Data Collection on Air Characters in MOT

Temperature distribution and RH data are experimentally measured in this study before furthering simulation of the MOT. The tools applied are summarized in table 3.

Table 3. Tools

No.	Material and Equipment
1	PVC Pipe
2	Pipe Stand
3	Thermocouple Wire (Type-T)
4	Data Logger
5	FLIR Ex-Series Infrared Cameras
6	Colemeter
7	Anemometer

For temperature distribution, the value is collected by using six panel of T-type thermocouple wire which four of the wire is label as T1, T2, T3, T4 and the other label as Inlet Temperature, T5 and Outlet Temperature, T6. Four labelled wire are assembled at the PVC pipe with stand and the inlet and outlet wire is placed at the inlet and outlet of the air conditioner. The value of temperature for each point is determined by using a data logger since all the wire is connected to it (figure 4). The pipe needs to be placed at each point in figure 5 for 10 minutes. At the same time, Colemeter which is the digital thermometer hygrometer humidity temperature sensor is used to measure the RH. This sensor is placed at the same point as in figure 6.



Figure 4. Channels Digital Handheld Temperature Thermometer Meter Data Logger

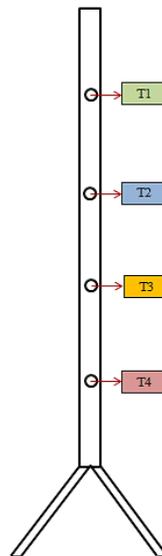


Figure 5. Four labelled wire in the PVC pipe

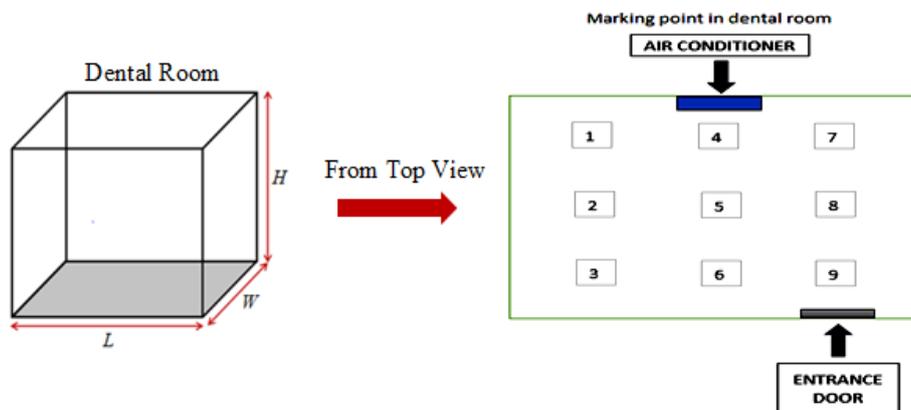


Figure 6. Marking point in MOT

For validation purpose on Temperature, RH and air flow velocity, the digital thermometer hygrometer humidity temperature sensor and the digital air flow anemometer with USB interface are applied (figure 7 and 8).



Figure 7. Digital thermometer hygrometer humidity temperature sensor



Figure 8. Digital air flow anemometer with USB interface.

3. Result and Discussions

Based on the experimental result as shown in table 4, the RH for MOT is 45.7% and the CFD Simulation for MOT is 47.1% respectively. The CFD approach was successfully performed to obtain the RH in MOT. The simulation results were related to the standard methods which are redefining the geometry of the MOT, optimization in mesh density, boundary conditions setting, solver setting, right solution methods and acceptable results were obtained from CFD post.

Figure 9 and 10 shows the CFD contour slices of the Mass Fraction of H₂O and Mass Fraction of Air inside Minor Operation Theatre. Based on the both figures (Figure 9 and 10). The ANSYS Fluent demonstrates 0.8% for Mass Fraction of H₂O and 99.2% for Mass Fraction of Air in MOT. Based on Total Mass-Weight Integral calculation, the RH is measured as 47.1%.

Table 4. Summary of results of Minor Operation Theatre

Details	Experiment Measurement of MOT	CFD Simulation of MOT
Relative Humidity %	45.7	47.1

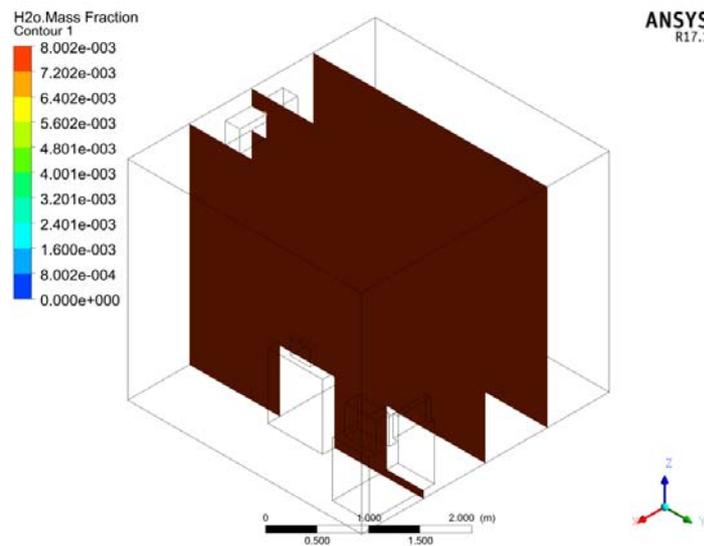


Figure 9. Slices of Mass Fraction of H₂O in Minor Operation Theatre based on location in figure 6.

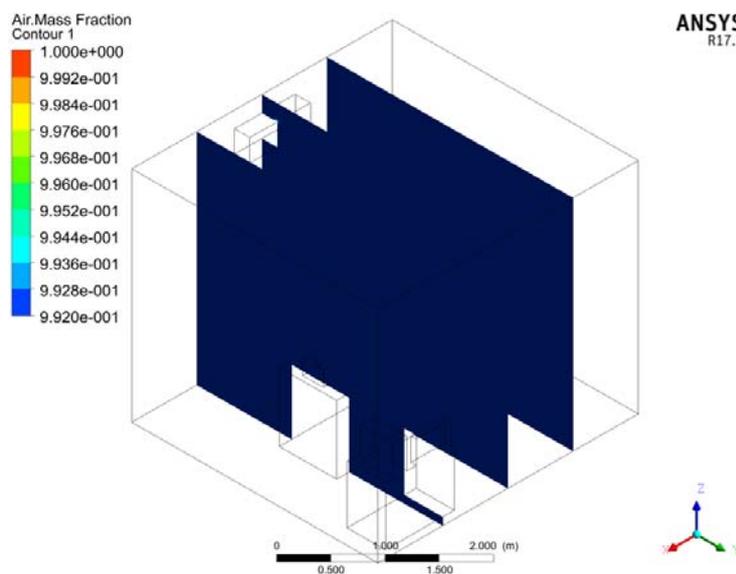


Figure 10. Slices of Mass Fraction of Air in Minor Operation Theatre based on location in figure 6.

4. Conclusion

The CFD simulation result for Minor Operation Theatre demonstrated differences in experimental result are about 3.06% for RH. 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 study observation, it reveals that ANSYS Fluent can be utilized for RH 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. Building 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.”