Proceedings of the 11th International Conference on Fluid Flow, Heat and Mass Transfer (FFHMT 2024)
Chestnut Conference Centre - University of Toronto, Toronto, Canada – June 16-18, 2024
Paper No. 040
DOI: 10.11159/ffhmt24.040

CFD of the Conditioned Air Distribution in a Hospital Operating Room

Omer E. Mohamed[1], Amr Ahmed[1,2], Musa Abubker[1,3]
[1]Department of Mechanical Engineering, University of Khartoum, Khartoum, Sudan
[1] omer.elfarouk@uofk.edu, [2] amraltijani@gmail.com, [3] musadirar415@gmail.com

Abstract
A numerical CFD simulation of an actual operating room in an educational hospital aims to determine the optimum interior air conditioning layout to achieve thermal comfort and contaminants removal from the operating room. The simulation investigates changing the location(s) and the size(s) of the supply air diffusers and the exhaust/return air grilles. The study examines four supply air diffusers and return air grilles' locations and sizes. The results reveal that the best locations are the central laminar air supply diffuser with two lower central exhaust/return air grilles.

Keywords**:** CFD, HVAC, Operating Rooms, IAQ

1. Introduction

A hygienic hospital operating room determines human life or death; thus, it needs particular concern. A study [1] shows that 5 to 10 percent of patients in acute care hospitals acquire one or more infections. This adverse event affects approximately 2 million patients annually in the United States, results in about 90,000 deaths, and adds an estimated $4.5 to $5.7 billion per year to the costs of patient care.

All the danger in operating rooms comes from the contaminants. Unfortunately, sterilization alone cannot remove it because of their propagation from the patient's wound. Many studies have shown that a very effective method to remove contaminants is driving them out by the conditioned air. Thus, operating rooms require ventilation and air conditioning to achieve thermal comfort and remove contaminants.

The design of an HVAC system for an operating room is built on many factors besides cooling load, beginning from the room structure, lights and surgeon's positions, equipment layout, and even the surgeon's movement. These factors make it hard to design an optimum system, and conducting experimental studies will be even more challenging because the work requires too many diffuser locations and sizes. Therefore, numerical simulation, which depends merely on computational fluid dynamics (CFD), is the more appropriate method for achieving this.

The increasing developments of computational fluid dynamics in recent years have opened the possibilities for improving HVAC systems in the design phase, with fewer experiments required, yielding low-cost yet effective systems [2]. One can apply CFD modelling and simulation to provide valuable indications on proper indoor microclimate conditions and IAQ (Indoor Air Quality) by examining the effectiveness and efficiency of various HVAC systems through quickly changing the location of diffusers, supply air conditions, and system control schedules [3].

According to the ASHRAE Applications Handbook [4], the temperature in the operating room (OR) should be in the range of 68–76F (20–24C), and the relative humidity should be between 50% and 60%, and these are semi-agreed with AIA guidelines (20-23C) and (45%-55%). ASHRAE and AIA state that positive air pressure should be maintained, and all air exhausted with no recirculation is preferred [5]. The NIH research has shown that 20 air changes per hour (ACH) are optimal for a general-purpose operating room. They sometimes specify higher air change rates for ORs where higher-risk procedures occur. Balocco et al. [3] confirmed the strong effects of a correct ventilation system design and location of the air supply diffusers on compliance with microclimatic conditions, IAQ levels, and satisfactory contaminant removal. Essam E. Khalil [6] recommended using a laminar diffuser as it achieved driving contaminants from the operating room. At the same time, Yunlong Liu et al. [7] asserted that operating with a 6-lamp light and a centre table under the laminar diffuser resulted in 100% particle displacement efficiency.

2. Mathematical Equations

Numerical simulation determines the efficient air conditioning system by solving the governing equations (in the discretized form) for the conservation of mass (continuity), momentum (Nervier-Stokes equations), energy, and species transport equations. In a Cartesian coordinate system.

Assuming that the flow is incompressible; thus, the mass conservation equation in the steady-state condition is as follows:

$$\nabla \cdot (V) = \frac{\partial}{\partial x}(u) + \frac{\partial}{\partial y}(v) + \frac{\partial}{\partial z}(w) = 0 \tag{1}$$

For incompressible flow, the general form of the momentum conservation equation is as follows:

$$\rho c_p \mathrm{u} \cdot \nabla T = k \nabla^2 T \tag{2}$$

Assuming that the thermal conductivity is scalar, with no heat generation, the simplified energy conservation equation becomes:

$$\nabla \cdot \left(\rho C_p T V\right) = \nabla \cdot (k \nabla T) \tag{3}$$

Assuming that the mass diffusivities of species in the airflow are scalars, thermal diffusion is negligible, and there is no chemical reaction, the species transport equation is given by:

$$\nabla \cdot (D \nabla C) + \nabla \cdot CV = 0 \tag{4}$$

Using ANSYS Fluent these equations are solved (in FVM discretization form) with the two realizable k-ε model equations which consists of kinetic energy equation and turbulent dissipation rate equation mentioned respectively:

$$\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_j}\left(\rho k u_j\right) = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right] + G_k + G_b - \rho\varepsilon - Y_M + S_k \tag{5}$$

$$\begin{aligned} \frac{\partial}{\partial t}(\rho \varepsilon) + \frac{\partial}{\partial x_j}\left(\rho \varepsilon u_j\right) &= \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_\varepsilon}\right)\frac{\partial \varepsilon}{\partial x_j}\right] \\ &+ \rho C_1 S \varepsilon - \rho C_2 \frac{\varepsilon^2}{k + \sqrt{v\varepsilon}} + C_{1\varepsilon}\frac{\varepsilon}{k} C_{3\varepsilon} G_b + S_\varepsilon \end{aligned} \tag{6}$$

The designer of OR requires rigorous work to achieve thermal comfort and contaminant removal. Inside the operating room, all fluid properties, including temperature, velocity, pressure, relative humidity, and contaminant removal, must be assessed using the mentioned governing equations for the various air conditioning schemes to determine the most efficient one.

3. Cooling Load Calculation

Table 1 illustrates objects dimensions and the heat fluxes inside the operating room. The velocity and temperature are found after estimating the cooling load (13.210 kW) using a psychrometric chart, as in Fig. 1. Noting that the patient's stomach, is considered a contaminant source.

Table 1: Objects dimensions and the heat fluxes inside the operating room

NO.	Entity	Temperature/ Heat flux	Dimensions (m)
1	Inlet	15 C	Variable
2	EDL Surgical lights(face)	210.122 W/m2	0.47 X 0.42 X 0.15
3	EDL Surgical lights(back)	10.506 W/m2	
4	Fluorescent lamps	200 W/m2	0.6 X 0.6
5	Wall	20.8274 W/m2	5 X 3
6	Roof	214.868 W/m2	5 x 5
7	Floor	29.05 W/m2	
8	Surgical unit	282.6 W/m2	0.46 X 0.51 X 1.01
9	Anaesthesia machine	14.02 W/m2	0.4 X 0.47 X 1.45
10	Surgeon	49.612 W/m2	0.25 X 0,25 X 1.55
11	Patient skin	91.263 W/m2	0.12 X 0.16 X 1.35
12	Patient wound	91.263 W/m2	

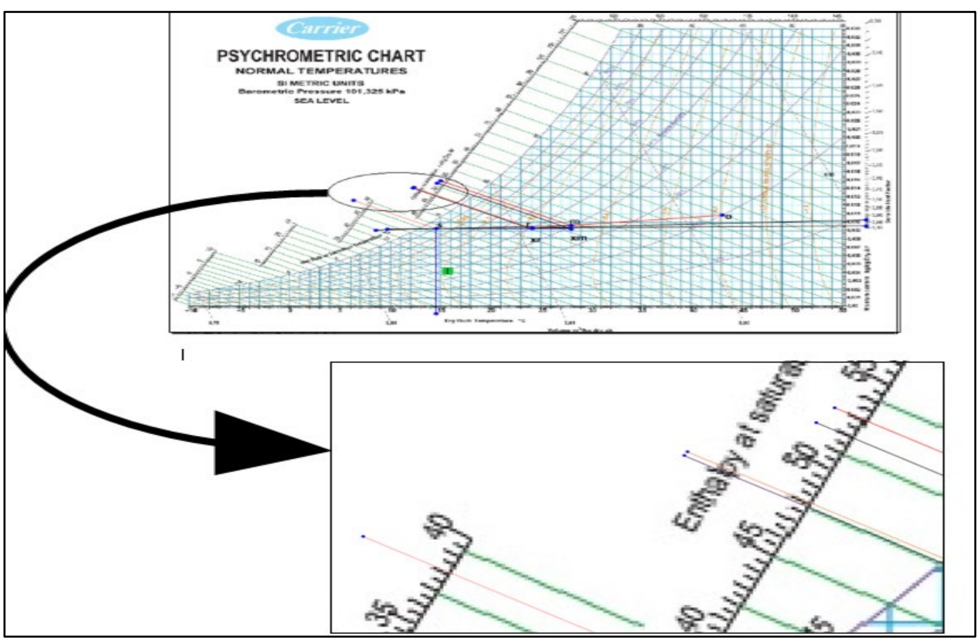

Fig. 1: Cooling load estimating using a psychrometric chart

4. Simulation Procedure

A room with dimensions 5m×5m×3m (illustrated in Fig. 2(a)) is simulated using ANSYS Fluent to achieve thermal comfort and contaminant removal. Due to the complexity of the geometry, which results in low-quality mesh, thus convergence problems, the geometry was simplified using symmetry plane which results in using less computational power. Fig. 2(b) shows the simplified model.

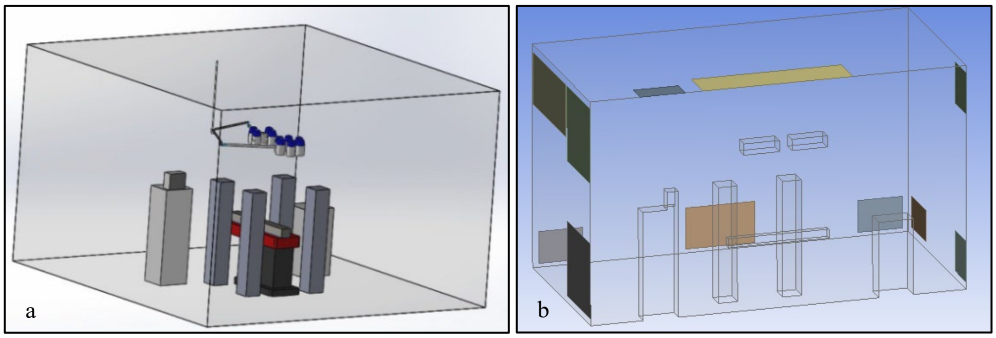

Fig. 2: (a) Basic Arrangement of the room, (b) Computational Model of the room

In the present case study, five meshes with different element sizes, 321,000, 620,000, 902,000, 1,350,000, and 2,048,000 elements, are investigated to identify the minimum mesh density to ensure that the converged solution obtained from CFD is independent of the mesh resolution. Velocity contours are used for comparing the performance of different mesh sizes by a horizontal line drawn across the room. The line location is selected in the most variant velocity contour in the y-direction (See Fig. 3(a)).

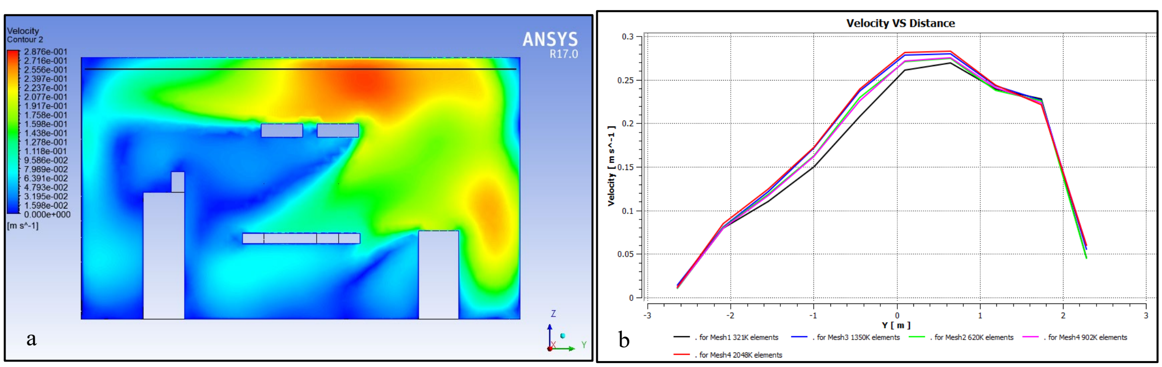

Fig. 3: (a) Mesh-independence study velocity line, (b) velocity results for different meshes (across the line)

Based on the above, a mesh with 1,350,000 elements (see Fig. 3(b) and Fig. (4)) is sufficient to carry out the simulations, as it gives a mesh-independent result at the minimum possible computational cost and time.

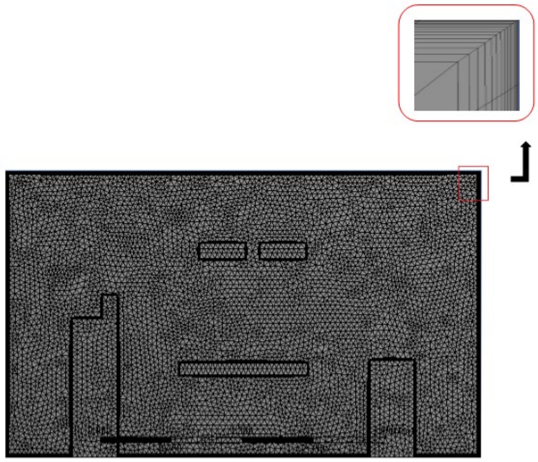

Fig. 4: The optimum mesh

In order to verify that the solution is insensitive to the error, three convergence criteria are used. The first one is mass balance which is tested by measuring the mass flow rate at both inlet and outlet and it's found that the value is equal for both (0.459 kg/s).

The other two criteria are Residuals stability and Average static temperature stability which are shown in Fig. 5(a), 5(b):

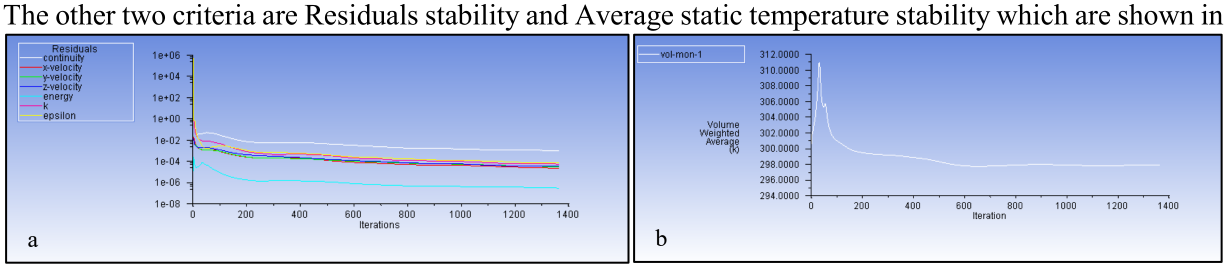

Fig. 5: (a) Residuals Vs. Iteration (Residuals stability), (b) Average static temperature Vs. Iteration (Temperature stability)

5. Validation of Results

The model and setup procedures are validated to obtain more accurate results by applying past studies conducted by Brazilian researchers [8]. They made experimental and numerical analyses of airflow in a surgical room. Their experimental study evaluated the environmental conditions (temperature and velocity) using four pedestals (points B, C, D, and E) with air temperature and velocity sensors positioned in the surgical room with various heights for each pedestal. They found good agreement between the simulation and the experiment. This study applied setup procedures in their case with their geometry and boundary conditions to compare the current numerical results (S2) with their numerical and experimental data (S1 and E1 respectively). The following table represents the comparison made:

Table 2: Velocity and temperature comparison at 4 points (each at 2 heights) between 3 cases (Validation of results)

Property	Point B			Point C			Point D			Point E		
	E1	S1	S2	E1	S1	S2	E1	S1	S2	E1	S1	S2
Velocity at 0.6 m	0.41	0.08	0.23	0.31	0.19	0.41	0.37	0.26	0.23	0.14	0.51	0.168
Velocity at 1.2 m	0.23	0.15	0.26	0.21	0.09	0.21	0.23	0.16	0.17	0.31	0.18	0.41
Temperature at 0.6 m	18.5	18.3	17.9	18.7	18.3	17.9	19.4	18.3	17.8	18.0	18.2	17.8
Temperature at 1.2 m	18.8	18.4	17.9	19.4	18.3	18	18.9	18.4	17.9	18.3	18.3	17.9

From table 2, it can be seen clearly that the new setup procedures are more accurate and robust than theirs because new results are so near to their experimental data, remarkably the velocity comparison.

6. Results and Discussion:

Two planes are located across surgeons and patients (see Fig. 6(a), 6(b)). Velocity vectors, temperature contours, contaminants concentration, and relative humidity are presented in these planes (as needed). Also, the mean values of previous characteristics in overall room (OA) and occupied zone (OZ) are presented in Table 4. Moreover, Cases details mentioned in table 3.

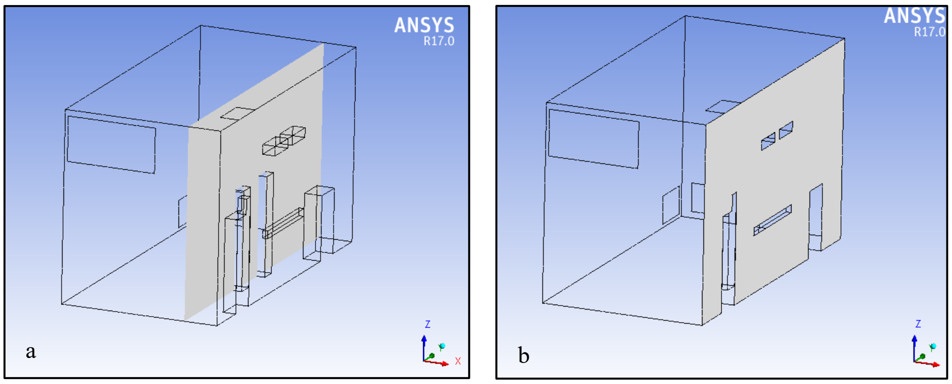

Fig. 6: (a) Plane 1 (drown across the surgeons, x=0.52 m from the plane of symmetry), (b) Plane 2 (drown across the patient, x=0m from the plane of symmetry)

Table 3: Air outlet(s) dimensions, velocity and arrangement

Case No:	Diffuser Area, (m^2)	Supply Velocity, (m/s)	Arrangement
1	0.7 X 1.4	0.3826	two upper-sidewall supply grilles, with two lower exhaust grilles in the opposite walls.
2	2 X 1	0.3826	one upper side wall (central horizontally) supply grille with one lower central exhaust grille on the opposite wall.
3	2 X 1	0.3826	one central supply diffuser in the ceiling, with two lower central exhaust grilles
4	2 X 1	0.3826	Case 4 represents one central supply diffuser in the ceiling, with four lower exhaust grilles

For case 1 and case 2, as seen (in Fig. 7(a), 8(a)) the cold air is concentrated near the right wall (due to the exhaust grille effect), so most of the conditioned air does not reach the occupied zone (of patient and surgeons) especially for case 1 at which the average velocity is 0.07 m/s and this results in forming circulations (especially in the left surgeon region) in which contaminants concentrated (CRE for occupied zone is found 0.21) (Fig. 7(b)). For Case 2 the CRE (Fig. 8(b)) and relative humidity is better (0.3 and 41.5% respectively) but still, most of the cold air exits without reaching the occupied zone (Fig. 8(a)).

In case 3 and case 4 the airflow is laminar, with low circulations formed, proved by the average velocity in the occupied zone (0.17 m/s and 0.16 m/s respectively) (Table 4) which falls in the laminar velocity range. So that the flow can wash more contaminants from the occupied zone and the average temperature in the occupied zone is about 20℃ for both cases, and it is noticed that the temperature at the head of the patient is a little relatively high (Fig. 9(a), 9(b)) due to the obstruction of the surgical lights. However, it is still normal (25°C).

The contaminant removal effectiveness (CRE) and average relative humidity for case 3 (0.4, 50.8 %) (Table 4) is better than case 4 (0.32,45%). It should be noted that case 3 has the preferable relative humidity [9].

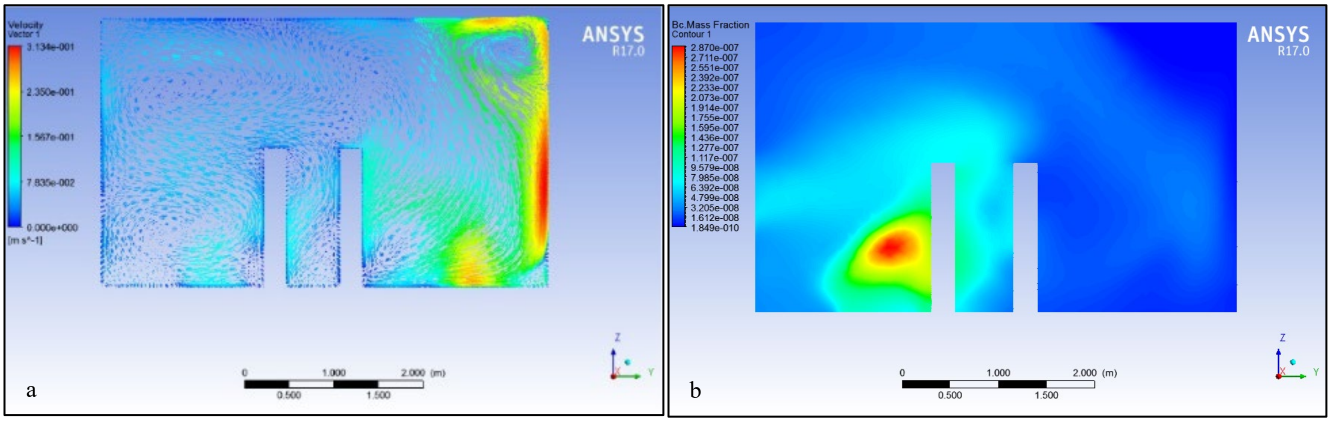

Fig. 7: (a) Velocity Vectors at Plane 1(case 1), (b) Contaminants Concentration at Plane 1(case 1)

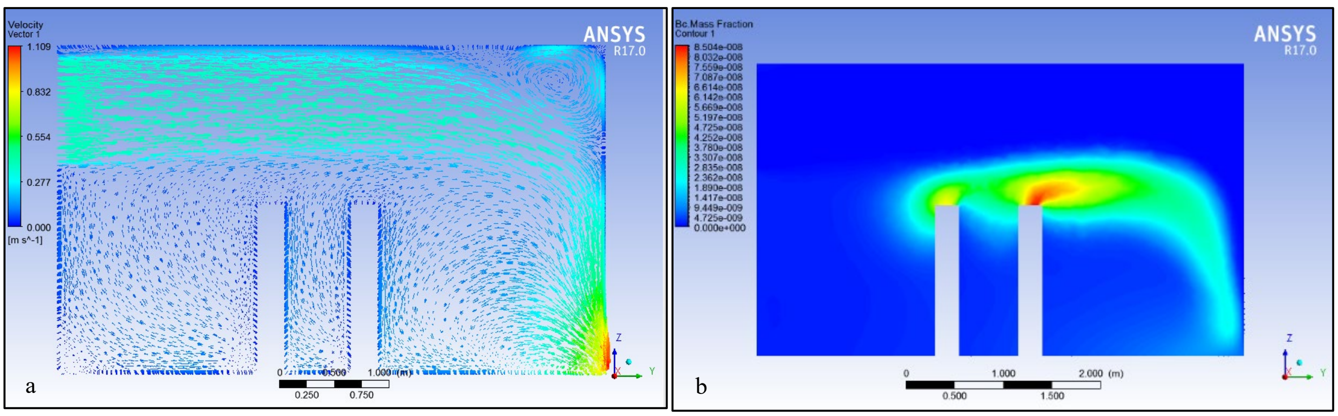

Fig. 8: (a) Velocity Vectors at Plane 1(case 2), (b) Contaminant Concentration at Plane 1(case 2)

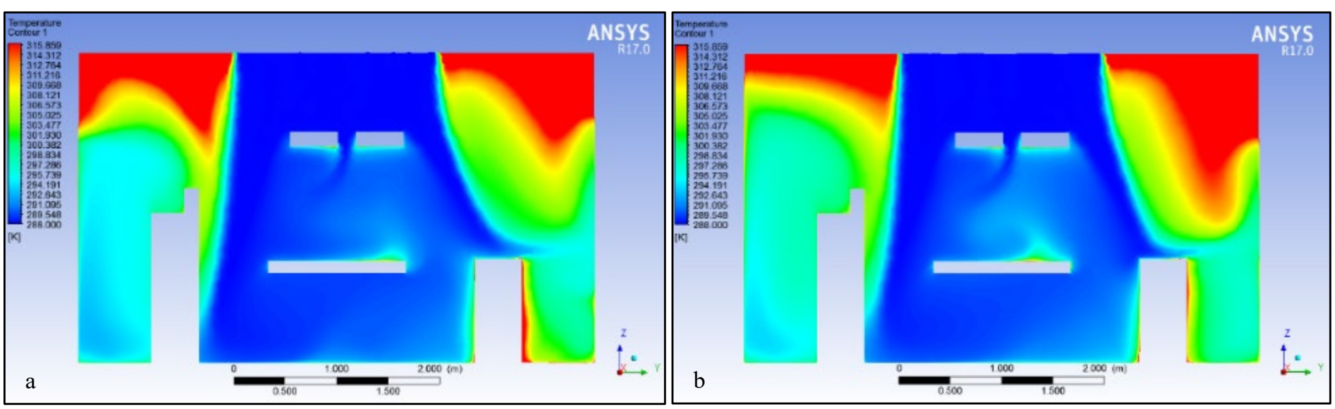

Fig. 9: (a) Temperature Contours at Plane 2(case 3), (b) Temperature Contours at Plane3(case 4)

Table 4: The Mean Characteristics in Multiple Zones

Case	Air Velocity, m/s		Temperature, ℃		Relative humidity, %	CRE
	OA	OZ	OA	OZ	OA	OZ
Case 1	0.14	0.07	30.9	36.9	38.5	0.21
Case 2	0.16	0.10	29.0	29.3	41.5	0.30
Case 3	0.12	0.17	30.1	19.9	50.8	0.40
Case 4	0.10	0.16	35.8	20.6	45.2	0.32

7. Conclusion

Numerical airflow, temperature distribution, and contaminant concentration simulations are carried out in an actual operating room. The results showed the strong effect of the supply diffuser(s) and outlet grille(s) position on thermal comfort and contaminant removal.

The central laminar diffuser with two central grilles near the floor (case 3) showed the best airflow, temperature distribution, and contaminants removal in the occupied zone, which is the concern, while the side wall diffusers (case 1 and case 2) didn't achieve thermal comfort because most of the conditioned air does not reach the occupied zone. Meanwhile, the central diffuser with four grilles near the floor (case 4) gave good results for thermal comfort, nearly the same as case 3, but in terms of contaminant removal, case 3 is better.

8. Recommendations

To obtain the best thermal comfort and contaminants removal in operating rooms, the following design considerations of the distribution system must be considered:

- The central laminar diffuser (located in the ceiling) has to have two central outlet grilles near the floor.
- The side wall diffuser (if used) must have an inclination angle.
- Further CFD work should be done to study the effect of surgical lights' position, the surgical staff's movement, the equipment layout, and the transient phenomena (e.g., door opening).

References

[1] Bill Drake, "Infection Control in Hospital", a supplement to ASHRAE Journal, June 2006

[2] Son H. Ho, Luis Rosario, Muhammad M. Rahman, "Three-dimensional analysis for hospital operating room thermal comfort and contaminant removal", Applied Thermal Engineering Journal, available online 13 November 2008 in journal homepage: www.elsevier.com/locate/apthermeng.

[3] Carla Balocco, Giuseppe Petrone, Giuliano Cammarata, "Numerical Investigation of Different Airflow Schemes in a Real Operating Theatre", Biomedical Science and Engineering Journal, Published Online February 2015 in SciRes. http://www.scirp.org/journal/jbise

[4] ASHRAE Applications Handbook, 2003

[5] AIA, AIA Guidelines for Design and Construction of Hospitals and Health Care Facilities, The American Institute of Architects, Washington, DC, 2006.

[6] Essam E. Khalil, "Energy Efficient Hospitals Air Conditioning Systems", Open Journal of Energy Efficiency 2012, Published Online June 2012 (http://www.SciRP.org/journal/ojee)

[7] Yunlong Liu, Alfred Moser, Kazuyoshi Harimoto, "Numerical Study of Airborne Particle Transport in an Operating Room", International Journal of Ventilation Volume 2 No.2, 2003

[8] Danilo de Moura, Victor Barbosa Felix, Marcelo Luiz Pereira, Arlindo Tribess, "EXPERIMENTAL AND NUMERICAL ANALYSIS OF AIRFLOW IN A SURGICAL ROOM" Copyright © 2007 by ABCM November 5-9, 2007, Brasília.

[9] McQuiston, Parker, Spilter, "Heating Ventilating, and Air Conditioning, Analysis, and Design", Six Edition