CFD Simulation of Air Pollution Distribution and Its Effect on Patients in Hospital Operating Rooms

Eiad Saif[1,*]
Abdulrazzak Akroot[2]
Abdul Raqib Abdo Asaad [3]

[1] Faculty of Engineering, Ondokuz Mayis University, Turkey
[2] Faculty of Engineering, Karabük University, Turkey
[3] Faculty of Engineering, University of Science and Technology, Yemen
* Corresponding author: eiad.saif@scc.edu.ye

https://doi.org/10.20428/JST.25.2.1

CFD Simulation of Air Pollution Distribution and Its Effect on Patients in Hospital Operating Rooms

Abstract:

This paper uses the CFD technique to simulate the internal environment of a hospital operating room, including the surgical area that includes the patient and the surgical team.

Moreover, this work provides mathematical simulation of many models to be placed in the air entry and air exit areas in the operating room to obtain the optimal model of air distribution within the operating room. The results obtained matched well with experimental data from approved literature and standards. These results found that airflow and air velocity significantly affect the patient’s thermal comfort in the operating room.Besides, it is observed that some recirculation zone is formed, and this is a result of the heat generated by the surgical team and the accelerated air. This region is considered a source for the survival of the polluted air in the room.

Keywords: CFD, Operating room, Numerical analysis.

المحاكاة بتقنية (CFD) لتوزيع تلوث الهواء في غرفة العمليات بالمستشفى

الملخص:

يستخدم هذا البحث تقنية تدفق السوائل الديناميكية (CFD) لمحاكاة البيئة الداخلية لغرفة عمليات مشفى، بما في ذلك المنطقة الجراحية التي تضم المريض والفريق الجراحي. علاوة على ذلك، يوفر هذا العمل محاكاه رياضيه للعديد من النماذج ليتم وضعها في مناطق دخول وخروج الهواء في غرفة العمليات للحصول على النموذج الأمثل لتوزيع الهواء داخل غرفة العمليات. النتائج التي تم الحصول عليها تتطابق بشكل جيد مع البيانات التجريبية من الدراسات السابقه والمعايير المعتمدة. وجدت هذه النتائج أن تدفق الهواء وسرعة الهواء يؤثران بشكل كبيرعلى الراحة الحرارية للمريض في غرفة العمليات. إلى جانب ذلك ، لوحظ أن بعض مناطق إعادة التدوير تتشكل، وهذا نتيجة للحرارة الناتجة عن الفريق الجراحي والهواء المتسارع. تعتبر هذه المنطقة مصدرًا لبقاء الهواء الملوث في الغرفة.

الكلمات المفتاحية : تقنية تدفق السوائل الديناميكية (CFD)، غرفة العمليات، التحليل العددي.

Highlights

• Both physical and mathematical CFD simulations were performed.

• The effects of velocity and temperature of entered air were proved.

• By changing the air velocity and temperature, thermal comfort in the room changes.

• The contribution of CFD applications in improving health outcomes was demonstrated.

1. Introduction

In recent years, the increasing development of computational fluid dynamics (CFD) has contributed to improving HVAC (Heating, Ventilation, and Air Conditioning.) systems at the design stage, and reducing the number of experiments requiredindoor air quality is an important part of studying, analyzing the indoor environment. The quality of the indoor environment is handled through the optimum control and adjustment of the technical features in HVAC systems. In a work environment such as hospitals, the quality of indoor air and thermal comfort directly affects patients' health and the well-being of the healthcare team [1-3].Consequently, HVAC systems should provide a comfortable working environment for health care providers in addition to minimizing risks, whether for health care workers or visitors [4-6].

Since most of the time people spend indoors, internal environments have to be studied and analyzed using the available techniques to ensure a comfortable environment [7-9]. In hospitals, the studying of this factor becomes mandatory due to many places founded that may be a source of infection and the spread of diseases. According to the CDC (center for disease control) reports, approximately 99, 000 deaths annually caused by healthcare-related infections. This statistic shows that hospitals are a suitable environment for infections. The surgical site infection is considered the most common in contributing to the high number of patient deaths, as infected skin squama's by staff are the main cause of infection [10-13].

In operating rooms, the risk is too much especially during the surgical operation because of the surgical area of a patient exposed to air. Many elements can cause post-operative infection, including factors of the patient, surgical space, operating room, and HVAC system. Also, the direction of

https://doi.org/10.20428/JST.25.2.1

airflow, and the position of the inlet, the outlet of air. Moreover, surgeons may become the main source of infectious particles and bacteria in an operating room for their activities [14-17]. Figure (1) shows most of the sources of infection in the operating room [11].

The spread of polluted air in hospitals is a very worrying issue and there is an obvious need for a system to distribute air effectively in hospitals [16]. Several studies have been conducted on CFD modeling for the spread of infection particles in indoor environments [14, 18, 19, 12, 20-22].

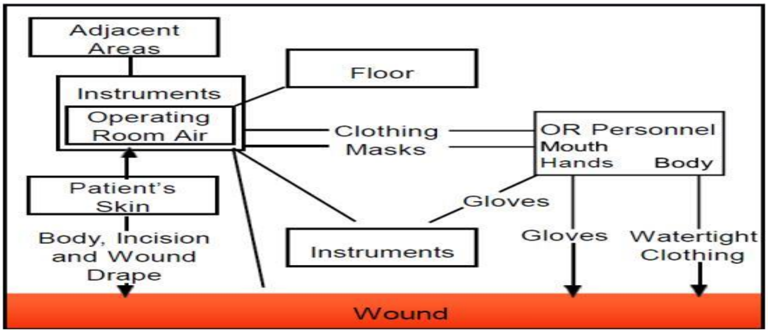

Figure 1: Sources of infection in the operating room [11]

In the African countryside, authors [23] studied a typical clinic by using CFD. The researchers pointed out that the research results can be used to make significant decisions to develop the health care system in the country. According to [24], it has been shown that the ventilation system and the type of airflow in indoor environments such as hospitals, could contribute to the transmission of infectious diseases. They recommended that the computational fluid dynamics (CFD) is a suitable predicting technique for air distributions and a preferred tool on hospitals operating room ventilation system designing.

In addition, [25] studied the energy performance of hospital buildings in the United Arab Emirates. The research indicated the need to consider the effects of thermal comfort conditions in future research because of its impact on patients in hospitals. The work [26] examined the effect of the ventilation system on infection control within the operating room by applying different ventilation options. The results of the experiment indicated the relationship between thermal comfort and the temperature of the air entering the room. Accordingly, this comfort can be maintained by controlling the air temperature. In another scientific study related to environmental controls in operating rooms [27], researchers have demonstrated that infection in operating rooms, particularly surgery sites, remains a major problem in recent medicine sciences.

Based on the aforementioned scientific research and given the importance of this research topic, this paper aims to study air velocity and temperature distribution in a hospital operating room and its impacts on patients and workers. In this research, the CFD modeling technique is used in the operating room to find the numerical solution for fluid flow and temperature, which develops a criterion that would help to improve ventilation efficiency.

2. MATHEMATICAL MODEL

In the study of indoor air quality and its effect, air parameters that affect airflows such as speed, temperature, and relative humidity should be determined. For airflow applications, the CFD tools find a solution for conservation equations for mass, momentum, and energy. Each of these equations includes the pressure, temperature, velocity of the element, and its surroundings, so it is best to solve the equations for all the elements simultaneously or consecutively.

The distributions of air flow were specified by solving the fluid flow equations. Navier Stokes, RANS, continuity and momentum equations, together with the energy equations.

2.1 Continuity and MomentumEquations

In the room model, the principle of mass continuity is shown more applicability, which presents as the control volume. Therefore, the time interval of the change of mass in the room must be identical to the difference between the mass entering and exiting the room as is evident in the following equation:

$$\partial\rho/\partial t + \nabla.(\rho\vec{v}) = 0 \qquad (3.1)$$

Considering ρ is the air density [kg / m3] and v is the velocity vector [m / s].

The Navier-Stokes equations define the basic mathematics of the motion of fluids, as they mainly represent Newton's law (Law of Motion) applied to liquids.

Based on Newton's second law of fluid flow, the momentum equation can be expressed in the following equation:

$$\nabla.(\rho\vec{v} \otimes \vec{v}) = \nabla.(\mu_{tot}\nabla\,\vec{v}) - \nabla p + \vec{F}_g + \vec{F}_{\Delta T} \qquad (3.2)$$

https://doi.org/10.20428/JST.25.2.1

Where μ_{tot} [kg/m. s] Where μ _tot [kg / m. s] represents the total molecular and turbulent viscosity, p is the pressure [Pa], and $\vec{F}_g + \vec{F}_{\Delta T}$ contain parameters such as thermal differences, $\rho\vec{v} \otimes \vec{v} = \rho\vec{v}.\vec{v}^T$ is the outer product.

The right side (Eq. 3) describes forces (body forces), while the left side describes time-dependent quantities such as acceleration.

2.2 Energy Equation

The energy in the air is defined as the sum of thermal energy (internal and velocity components) and the gravitational potential energy in buoyancy-driven flows. Consequently, the energy can be maintained in a stable state as follows:

$$\nabla.(\vec{v}(\rho E + p)) = -\nabla.\left(\sum_j h_j J_j\right) + S_h \tag{3.3}$$

Where J_j is the diffusion flux of species j, h_j is the enthalpy of species j, and S_h includes the heat or any other volumetric heat sources defined in the simulation process.

2.3 Turbulence Equation

To numerically simulate the turbulent flow, it was modeled with the k-ε turbulence model. Depending on the instantaneous velocity u (x, t) of the flow at position x and time t. The instantaneous velocity u_i can be described at a specific location and time:

$$u_i = \bar{u}_i + u'_i \tag{3.4}$$

$\bar{u}_i$ can be described asanaverage of v for stable flow, and u'_i is fluctuation velocity. u'_i is obtained by measuring as the standard aeviation of u_i. Turbulence intensity (TI) can be calculated as:

$$TI \equiv u_i/\bar{u}_i \times 100 \ \ inprecent \tag{3.5}$$

The turbulent kinetic energy k is given per unit mass as:

$$k = \frac{1}{2}\bar{u}_i'^2 = \frac{1}{2}\left(u_1'^2 + u_2'^2 + u_3'^2\right) \tag{3.6}$$

3. CFD MODELING

Computer simulations can contribute to obtaining higher accuracy of results, thus simplifying and improving the research process.The most important advantage of CFDs is the ability to test various system configurations (types of models, positions of equipment, etc.) virtually without creating real systems.

In this way, it is possible to optimize the suitable design of the ventilation system and adequate locations of air inlets/outlets to decrease the sources of infections in an operating room by numerical methods.

3.1 Model Establishment

The first step is to perform a CFD analysis for a fluid region of interest is drawing and meshing in specific geometric modeling using the ANSYS Design Modeler application. Figure (2) shows a simplified three-dimensional model of the typical operating room, in terms of the number of surgical staff, surgical lights, and operating table. In addition to taking into account the air inlets and exits on the sidewalls for the basic simulation model.

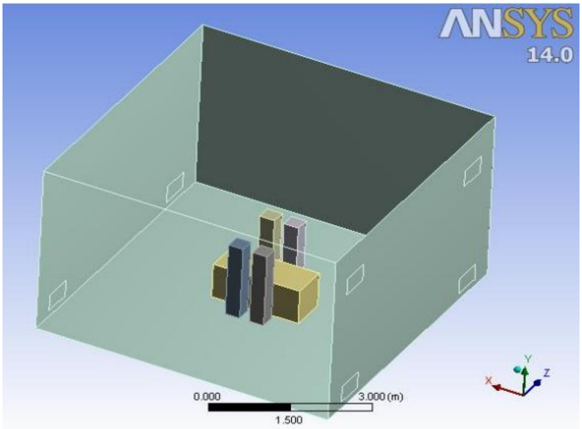

Figure 2: Simplified Typical Operating Room [21]

The simulation processes were performed using four tools of ANSYS. These tools start with the Design Modeler followed by Meshing and thenFluent and CFD-Posttool. In one interface, these tools allow users to perform the simulations.

4. RESULTS

The results obtained from this work were evaluated by comparison of the air temperature and air velocity with the numerical results presented in the references[28, 1, 29].Table 1 illustrates the results reached in this work and the numerical data presented in the literature. The comparison showed a compatible between them, which mentions an agreeable verification of the current work.

Table 1. A comparison of results obtained from this work with data from [29 ,28 ,1].

	Present Work	Ref. [28]	Ref. [29]	Ref. [1]
Air temperature	20 ° C - 22 ° C	22 ° C - 24 ° C	17 ° C - 22 ° C	14 ° C - 29 ° C
Air velocity	0.35 – 0.4 m / s	0.127-0.178 m / s	0.45±0.1 m / s	0.46 m / s or below

Figure (3) shows the contours of temperature in a plane 1(z-x plane) with height 2.8 m from the floor and plane 2 (z-x plane) with height 1.7 m from the floor. The neighboring area next to the inlets of air supply generates regions with a low temperature, where the air enters the operating room is cold.

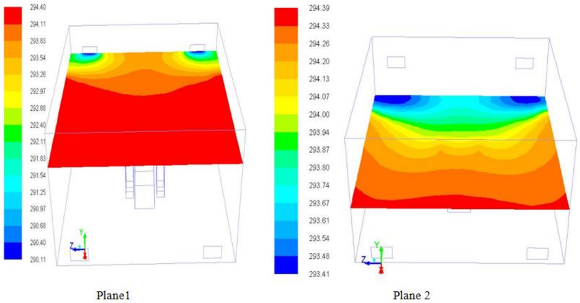

Figure 3: Temperature contours with location of plane 1 and plane 2

To more investigation, the distribution of temperature near ahead of the patient was plotted on the (x-y) axis as shown in Figure 4. In the lower region, the temperature is decreased as the air cold enters the room near this site.

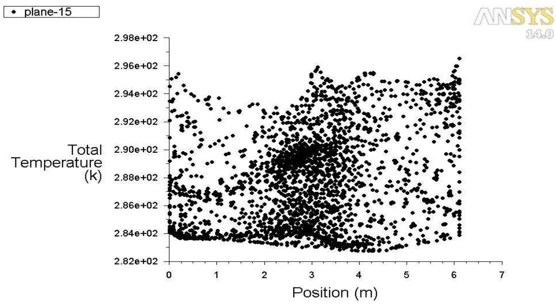

Figure 4: Temperature distribution beside the patient area

Concerning the study of air inside the room and its impact on the surgical area, the flow of airflows to the room at a speed of (0.4 m / s) through the supply hole on the right wall. Figure (5) presents the iso-surface cut in the middle of the surgical site. It can be seen that the average air velocity at the center of the surgical site is less than 0.35 m/s.

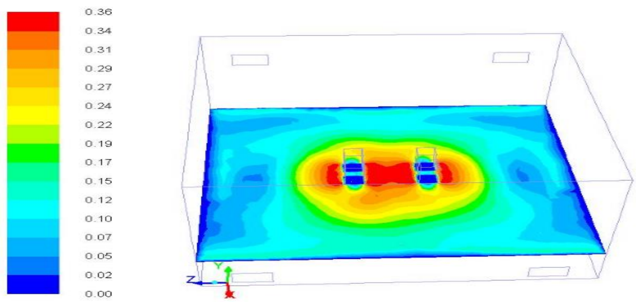

Figure 5: air distribution on the middle of surgical site y=1m

For further investigation, the velocity field is mapped on the filled velocity vectors as shown in Figure 6. The upper and lower flow created a circulation area due to high temperature and low-temperature surfaces. This flow is mixed with hot air coming out of the surgical site, extending along with the ceiling, falling at the wall, and then coming out of the low exits (3, 4).

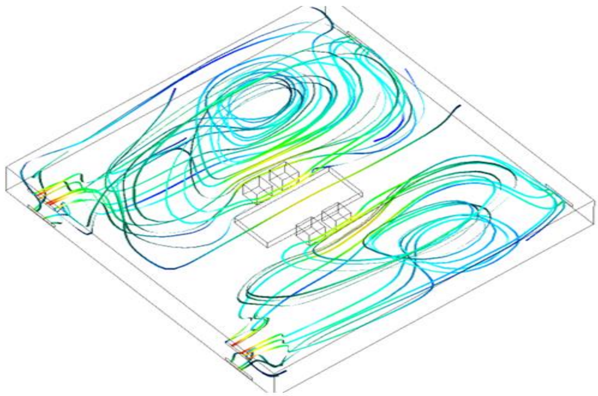

Figure 6: Streamline Contours within all the space of the OR

5. DISCUSSION

5.1 Air Temperature

Through simulation results, in which case the air is flowing into the operating rooms through the entrances (1, 2) at the top of the right wall, at maximum speed (0.4 m / s). Concerning the distribution of air temperature as one of the factors of study in this research, it has been found that the different parts of the indoor temperature difference are formed. it was found that the air with low intensity (warm air) goes up and make region with high temperature,

https://doi.org/10.20428/JST.25.2.1

but the air with high intensity (cold air) goes down and make region with low temperature. Compared with the standard internal temperature, the results are acceptable and there are some slight increases or decreases in some casesbut in the same allowed ranges.

5.2 Air Velocity

In respect with studying the air inside the room and its effect on the surgical area, by maintaining the same location and speed of the air entering the room, it was noted that its speed decreased at the site of the surgery to (0.35 m / s) and yet it is at the acceptable standard level and the patients and surgical staff will feel more comfortable. Accordingly, it can be said that the air velocity (≥0.3 m/s as inlet value) contributes to reducing the recycling areas near the operating table.

On the other hand, when observing the air velocity vector as streamline from the entry area through the surgical area, it is observed that some recirculation zone is formed, and this is a result of the heat generated by the surgical team and the accelerated air. This region is considered a source for the survival of the polluted air in the room.

6. CONCLUSIONS AND FUTURE WORKS

In this paper, CFD analysis of airflow, temperature distribution, is presented and discussed in a 3D hospital operating room with conventional airflow. The simulation results show acceptable results compared with experimental data from the literature and standards. This work provides a comprehensive analysis of air velocity and temperature distributions in the operating room. Based on this analysis it was found that airflow and air velocity significantly affect the thermal comfort of all operating theater personnel.Therefore, the air that enters the room is very important and affects the room temperature. Air distribution at the wound site indicates that air velocity also within the required ranges. Besides, the average temperature at the surgical site is between 20 ° C and 22 ° C, so that these values fall within the recommended range of 14 ° C to 29 ° C.

From the present observations, it can be concluded that the application of CFD in hospitals will improve the design of the internal environment as well as health outcomes.

In future works, more researches are needed on the effect of radiant heat transfer between the interior surfaces of the room and heat transfer across the walls as well as all the equipment in the room. In addition to applying the model practically in a real operating room and make a comparison between the simulation and experimental results.

References

[1] Balaras, C.A., et al., 2007. HVAC *and indoor thermal conditions in hospital operating rooms.* Journal of Energy and Buildings. 39(4): p. 454-470.

[2] Dascalaki, E.G., et al., 2009. *Indoor environmental quality in Hellenic hospital operating rooms*. Journal of Energy and Buildings 41(5): p. 551-560.

[3] Leung, M. and A.H. Chan, 2006. *Control and management of hospital indoor air quality.* Med Sci Monit. 12(3): p. SR17-SR23.

[4] Liu, J., et al., 2009. *Numerical simulation on a horizontal airflow for airborne particles control in hospital operating room.* Journal of Building and Environment. 44(11): p. 2284-2289.

[5] Peri, A., et al., 2011. *Numerical simulation of air flow in a general ward of a hospital*. IJRRAS. 8(3): p. 400-444.

[6] Rui, Z., et al., 2008. Study on biological contaminant control strategies under *different ventilation models in hospital operating room*. Building and Environment. 43(5): p. 793-803.

[7] Balocco, C. and P. Liò, 2010. *Modelling infection spreading control in a hospital isolation room*. JBiSE. 3(07): p. 653.

[8] Hamdani, M., et al., 2017. *The Study Natural Ventilation by Using Buildings Windows: Case Study in a Hot Dry Climate, Ghardaïa, Algeria.* Journal of Energy Procedia. 139: p. 475-480.

[9] Yin, S., et al., 2019. *Transient CFD modelling of space-time evolution of dust pollutants and air-curtain generator position during tunneling*. Journal of Cleaner Production. 239: p. 117924.

[10] Cristina, M., et al., 2016. *Operating room environment and surgical site infections in arthroplasty procedures.* Journal of Preventive Medicine Hygiene. 57(3): p. E142.

[11] Memarzadeh, F. and A.P. Manning, 2002. *Comparison of operating room ventilation systems in the protection of the surgical site/Discussion*. ASHRAE transactions. 108: p. 3.

[12] Sadrizadeh, S. and S. Holmberg, 2014. *Surgical clothing systems in laminar airflow operating room: a numerical assessment.* Journal of infection public health. 7(6): p. 508-516.

[13] Van Gaever, R., et al., 2014. *Thermal comfort of the surgical staff in the operating room.* Journal of Building Environment. 81: p. 37-41.

[14] Chow, T.-T. and J. Wang, 2012. *Dynamic simulation on impact of surgeon bending movement on bacteria-carrying particles distribution in operating theatre.* Journal of Building Environment. 57: p. 68-80.

[15] Kang, Z., et al., 2017. *Numerical Analysis of Ward's Flow Field and Pollutant Distribution and Its Impact of Patients and Visitors.* Journal of Procedia Engineering. 205: p. 4122-4128.

[16] Nielsen, P.V., et al., 2010. *Risk of cross-infection in a hospital ward with downward ventilation.* Journal of Building Environment. 45(9): p. 2008-2014.

[17] Satheesan, M.K., et al., 2020. *A numerical study of ventilation strategies for infection risk mitigation in general inpatient wards. in Building simulation.* Springer.

[18] Hathway, E., et al., 2011. CFD *simulation of airborne pathogen transport due to human activities.* Journal of Building Environment. 46(12): p. 2500-2511.

[19] Murakami, S., et al., 2003. *Modeling and CFD prediction for diffusion and adsorption within room with various adsorption isotherms*. Journal of Indoor Air. 13: p. 20-27.

[20] Saif, E., 2015. *A Study of Quality and Distribution of Air in Operating Rooms Using Computational Fluid Dynamic (CFD) Technique* 2015, University of Science & Technology.

[21] Saif, E., et al., 2016. *Numerical Simulation on Conventional Ventilation System for Hospital Operating Room*. Journal of Science Technology. 21(1).

[22] *Sidawi, B., et al., 2014. CFD modeling as a tool for assessing outdoor thermal comfort conditions in urban settings in hot arid climates.* Journal of Information Technology in Construction.

[23] Khalid, A. and C. Scherrer, 2013. *Alternatives for Reducing the Risk of Transmission of Tuberculosis in a Typical Hospital Clinic in Developing African Countries.* Journal of Procedia Computer Science. 16: p. 853-862.

[24] Villafruela, J.M., et al., 2013. *Comparison of air change efficiency, contaminant removal effectiveness and infection risk as IAQ indices in isolation rooms.* Journal of Energy Buildings. 57: p. 210-219.

[25] Taleb, H.M., 2016. *Enhancing the skin performance of hospital buildings in the UAE.* Journal of Building Engineering. 7: p. 300-311.

[26] Al-Waked, R., 2010. *Effect of ventilation strategies on infection control inside operating theatres.* Journal of Engineering Applications of Computational Fluid Mechanics. 4(1): p. 1-16.

[27] Dharan, S. and D. Pittet, 2002. *Environmental controls in operating theatres.* Journal of Hospital Infection. 51(2): p. 79-84.

[28] ASHRAEA, 2006. *Ventilation of Health Care Facilities.* American Society for Heating, Refrigerating Air-Conditioning Engineers., ed. G. Inc.: Atlanta, USA.

[29] Handbook, A.F.J.I.A., GA, USA, 2009. *American society of heating, refrigerating and air-conditioning engineers.*