International Journal of Engineering

Design of Air Distribution System for Operation Theatre Using Flow Visualization Techniques to Improve Flow Characteristics

S. D. Rahate*, A. D. Sarode

Department of Mechanical Engineering, Lokmanya Tilak College of Engineering, University of Mumbai, Navi-Mumbai, India

P A P E R I N F O	A B S T R A C T

Operation Theatre (OT) is the most important area where precise controlled on indoor air quality is required because it is directly related to patient health and its recovery period. This work aims to study and visualize the airflow distribution of conventional flat air diffuser ventilation system and newly designed angular air ventilation system in OT. Angular Air Distribution (AAD) system is designed in such way that, conditioned air is throw in OT with some angle so it can cover the maximum area of OT with laminar flow and minimum installation area. The performance of angular air distribution (AAD) system with varying inlet diffuser angle, constant 0.4m/s inlet velocity and air flow pattern are investigated with the help of numerical simulation and experimental setup of prototype OT model. Detail experimental investigation is done on a prototype with the help of a smoke test and camera target method (CTM). Both numerical and experimental results showed a similar pattern of air distribution for the various design configurations.

1. INTRODUCTION

The main task of Heat Ventilation and Air Conditioning (HVAC) system in the modern hospital Operation Theatre (OT) is to provide a comfortable and healthy environment to the patient and surgeon. As ventilation system play important role in HVAC to keep OT environment free from bacteria. The poorly ventilated system not only affect the health of patient and surgeon from infection but also feel them uncomfortable [1]. In previous standard and guidelines, turbulent air flow was allowed in OT ,but recently published standards suggest to use Laminar Air Flow (LAF) systems [2]. Many national and international standards and guidelines are available for every country to design a ventilation system with environmental parameters for OT [3-6]. Alternate methods are mixing and displacement ventilation system which differs by the position of supply and an exhaust port in OT. Indoor obstacles can easily affect the unidirectional flow pattern of the ventilation system. Ventilation and air distribution pattern have a great effect on indoor air quality which includes indoor temperature,

relative humidity, air flow velocity, pressure relationship, air movement's efficiency. Therofore in this work new AAD system is designed in such a way that it will distribute the air in laminar pattern in whole OT. As per national and international standards and guidelines, the temperature in OT should be maintained at 18-22ºC with minimum temperature draft because it affects human comfort. Relative humidity should be maintained 45 – 55% because high humidity causes thermal discomfort and low humidity results in blood coagulation. The laminar air flow is a common to design for OT with air velocity 0.2-0.4 m/s to maintain unidirectional airflow pattern in the critical zone [7]. Balaras, C.A et al. [8] investigated 20 OTs in 10 different hospitals and recorded information on the type of ventilation system, temperature range, relative humidity range, etc. Chow and Yang [10] had investigated that, thermal comfort is achieved by controlling the temperature, the humidity, and air movement. In laminar air flow, supply is given at the rate of 0.46 m/s ,there are two types of LAF one is horizontal and vertical. In this literature velocity is measured by using hot wire anemometer and compared it

Corresponding Author Email: swati.rahate363@gmail.com (S. D. Rahate)

with a mathematical k-e model for validation purpose [9-11]. Sasan Sadrizadeh et al. [12] investigated the vertical and horizontal ventilation system in terms of sedimentation and bacteria distribution in operation theatre. He used the CFD software for numerical calculation and validate it by comparison with experimental data reported in the literature. Liu et al. [13] investigated an alternative of horizontal flow pattern and air flow performance in an OT. He also evaluates the effectiveness of the horizontal unidirectional airflow to control infectious airborne particles in operation theatre. In this study, the above drawbacks which is observed in literature is overcome by designing Angular Air Distribution (AAD) system. In AAD system, conditioned air is throw in OT with some angle so it can cover the maximum area of OT with laminar flow and minimum installation area. The designed AAD ventilation system is examined with the help of numerical simulation by varying inlet angles at , constant value and 0.4m/s inlet velocity. It is validated by experimental measurement and observations performed on prototype OT model. The air flow pattern of AAD system by varying inlet angles were tested with the help of smoke test. The air flow pattern obtained by the smoke of AAD system is captured by Camera Target Method (CTM) and it is validated with CFD contours plot results.air condition requirement

2. MATERIALS AND METHODS

2. 1. Numerical Model   The 3D model of prototype OR is modeled in CATIAV5 modeling software as shown in Figure 1. The OT room geometry that outline by a rectangular shaped room of wooden material with 1.524 m (L) x 1.524 m (B) x 0.9 m (H). Square ceiling supply diffuser (0.25 m^2 surface area) and Four exhausts (0.00580 m^2 cross-section area) are arranged over two of the opposite side wall. The operating bed is located at the middle area of OT under laminar flow. The five surgical staff each standing around the surgical table. They were modelled as cylinders. To investigate turbulence formation in air flow, Reynolds Averaged Navier Stokes (RANS) equations were employed to simulate the air flow field. It has been reported in the literature [11, 12], that the two equation RNG k-epsilon model is proper for indoor airflow simulation. This model was developed using renormalization. K-epsilon (k-ε) turbulence model is the most common model used in CFD to simulate mean flow characteristics for turbulence flow conditions [14, 15].

Conservation of mass equation:

$$\nabla \left(\rho \vec{v} \right) = S_m.$$ (1)

Momentum:

$$\nabla . \left(\rho \vec{v} \vec{v} \right) = -\nabla p \; + \; \nabla . (\bar{\bar{\tau}}) + \rho \vec{g}$$ (2)
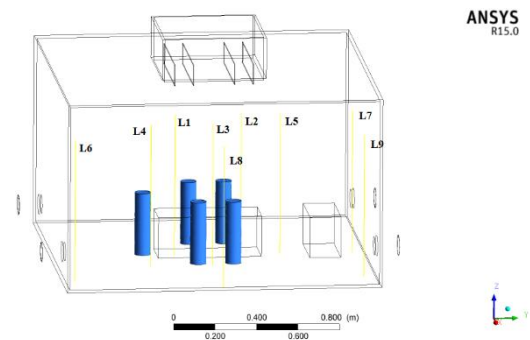

Figure 1. Solid model of OT with interior features and inlet-outlets

Where $\bar{\bar{\tau}}$ is viscous stress and can be written:

$$\bar{\bar{\tau}} = \mu \left[(\nabla \vec{v} \; + \; \nabla \vec{v}^T) \; - \frac{2}{3} \; \nabla . \vec{v} I \right]$$ (3)

Energyconservation:

$$\nabla . (\vec{v} \left(\rho \vec{v} \; + p \right)) = -\nabla . \left[\sum_j h_j J_j \right] \; + \; S_h .$$ (4)

Turbulence Kinetic Energy (k):

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

Dissipation Rate of turbulence kinetic energy (ε)

$$\frac{\partial}{\partial x_i} (\rho \varepsilon u_i) = \frac{\partial}{\partial x_j} \left[\left(\mu + \frac{\mu_t}{\sigma_\varepsilon} \right) \frac{\partial \varepsilon}{\partial x_j} \right] C_{1\varepsilon} \frac{\epsilon}{k} \left(G_k + G_{3\varepsilon} G_b \right) - C_{2\varepsilon} \rho \frac{\varepsilon^2}{k} + S_\varepsilon$$ (6)

$C_\mu , C_{1\varepsilon}, C_{2\varepsilon} \; \sigma_k , \sigma_\varepsilon$ are constant of turbulence model and their values are $C_\mu = 0.09$, $C_{1\varepsilon} = 1.44$, $C_{2\varepsilon} = 1.92$, $\sigma_k = 1.0$, $\sigma_\varepsilon = 1.3$

∇ is partialliy derivatives of quantity in all direction G_k, G_b is generation of turbulence kinetic energy due to the mean velocity gradient, S_h and S_m is strain tensor with respect heat dissipation, S_E, S_k is user defined source term obtained by K and ε model equation, S_{ij} is strain Tensor in I and j direction and respective modulus and buoyancy. The values of physical properties of material used in numerical simulation are listed in Table 1.

Figure 2 shows the mesh model of the OT. The different mesh size is considered for mesh independence study. Tetrahedron shaped cells were used with a varying number of elements of 256916, 287258, 312761, 39844, 586479. The velocity plot for different mesh size is

TABLE 1. Physical Properties of material used in numerical simulation

Material	ρ (kg/m^3)	C_p (j/kg k)	K (w/mk)
Wood	700	2310	0.173
Air	1.225	1006	0.02435

plotted and it was approximately same for last four number of elements.The criteria for converged numerical solution are taken as 1e-3 RMS with element size 586479as shown in Figure 3.

Figure 3 shows the changes in velocity profile with OT height according to mesh element size. An inlet velocity is taken as 0.4 m/s with inside pressure as atmospheric. In this work, Ansys-Fluent 15.0 was used to solve the momentum, continuity and turbulence equations for fluid flow [15]. Since fluid flow is laminar and turbulence, standard wall function and turbulence k-e model is used for analysis. The simulation was conducted under steady state condition. The number iteration is taken for selected K-epsilon model and solution are converged. Inlet velocity is taken as 0.4 m/s ,as inlet boundary condition with 5% turbulence intensity for a hydraulic diameter of OT. The exhaust port 1.3014 Pa pressure-outlet boundaries with no-slip boundary condition are combined with zero heat flux to define adiabatic walls.The walls of OT is considered as standard wall function including interior of OT.

2. 2. Experimental Setup   The general layout of the prototype model of OR with angular air distribution system is shown in Figure 4. It consists of two main parts such as air distribution system i.e. supply diffuser and interior. The OT prototype room model
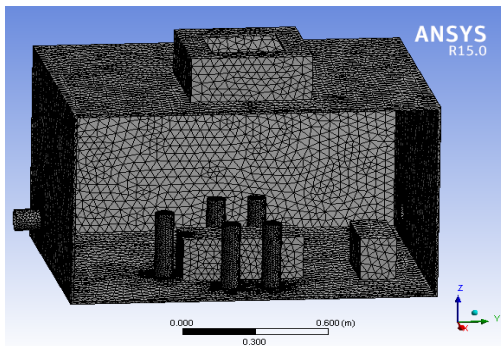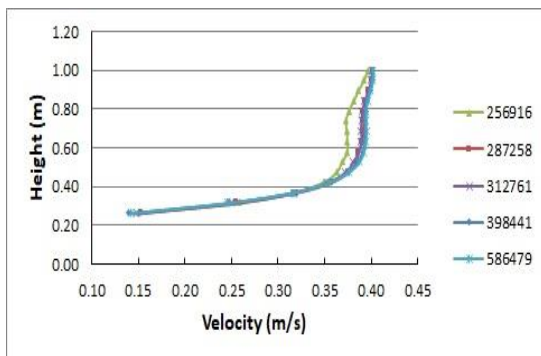

Figure 2. Mesh model of OT

Figure 3. Velocity profile for different element size

made of wooden material. Which scale by 1:4 ratio of full scale of prototype.

In order to validate the results obtained from the CFD solution, a scaled model of OT is tested by experimental measurement i.e smoke test with help of CTM. For clear visualization of airflow pattern inside the OT, the interior is painted as black and white coloured smoke is used with air. A smoke generator liberates smoke which is accumulated in the mixing box. It is connected to inlet diffuser of OT by means of flexible ducts. The quantity and speed of the smoke are controlled by two valves which are placed on inlet and outlet port of the mixing box. The smoke entered in OT through inlet diffuser and exhausted to the atmosphere by outlet port which is placed at the bottom of the walls.Velocity is measured by temperature compensate Hot wire anemometer (Measuring Range: - NTC: -20 to 60 deg C, Accuracy: - $\pm$ (0.1 m/s + 5 % of mv) (0 to 2 m/s)) is measured inside the OT, Diffuser inlet and AHU outlet. To access all 27 points by annemometre probe, the length of probe is 3 meter long to measure the velocity at middle plane of OT. The velocity is measured on 9 lines fixture which is fixed at different location inside OT. The line fixture fixed with 3 measuring point at different height 200, 500, 700 mm from the floor (27 points) inside the OT, side wall of scale model is drilled with the hole to insert the probe of anemometer at different point as shown in Figure 1. AAD system is installed on ceiling at middle area of OT. AAD is square ceiling supply diffuser with 4 number of louvers equally spaced from edged of the diffuser is located in the central zone of the ceiling, that guide the unidirectional and angular flow by varying angle of louvers as shown in Figure 5.The aim of the test is to observe the airflow pattern and its distribution throughout the OT. The front side of the prototype is kept transparent so that clear images can be captured. The camera is fixed at its position using a tripod and the smoke is fed to flow inside the prototype. For smoke visualization test, smoke generator (Power: - 900watt, heat up time -12 min, capacity-3 liters) with fog fluid (white cloud) is used.
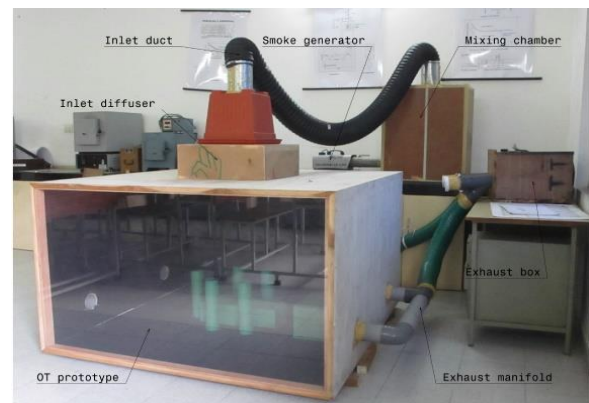

Figure 4. Experimental set up for smoke visualization in OT

The smoke is stored in the mixing chamber to reduce the velocity, mixing chamber act as an accumulator, and the flow rate of smoke is controlled by valves. The interior features of OT and the white smoke is targeted and its images are captured by high definition digital camera. Initially, smoke visualization test is carried out to ensure no-leakage in the room. Further smoke visualization test was carried out to identify the air flow pattern with different inlet angle of AAD system and to evaluate the effectiveness of the same system.

3. RESULTS AND DISCUSSION

The velocity measurements and air flow pattern were taken from the ceiling to the floor on middle plane of scale model OT with constant velocity 0.4 m/s and outlet pressure atmospheric. The separate CFD and experimental analysis are taken for 3 cases of AAD system with 90°, 60°, 45° angle of louvers.

From Figures 6-8 the air stream after striking the bed gets reflected upward causing the vortex. This vortex spread over the OT and mixes with fresh incoming primary airflow. These vortexes cause temperature draft (Canada effect) and turbulence in the flow of harmful particles along with the airstream flow over the surgeons and remain in the critical zone.

From Figures 9-11. it is observed that air velocity in critical zone is varying from 0.2-0.3 m/s, which is in
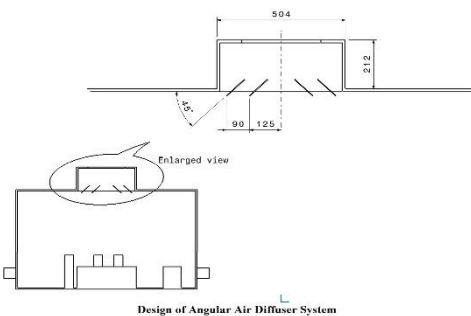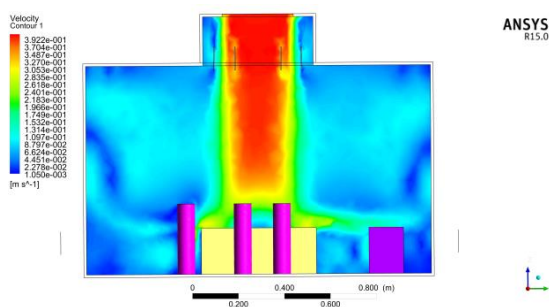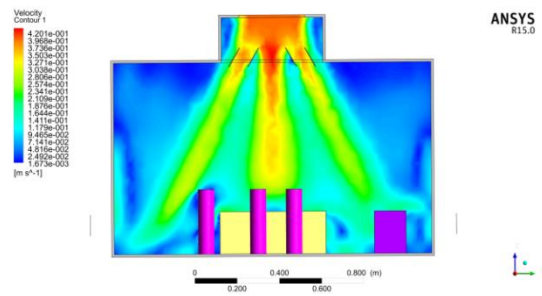

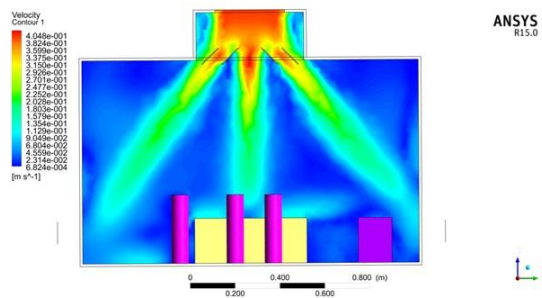**Figure 7.** Velocity Plot for OT with 60° Angular diffuser

Figure 8. Velocity Plot for OT with 45° Angular diffuser

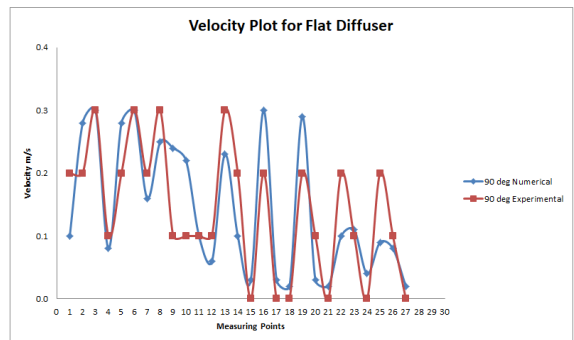acceptable range as mentioned in national and international standards and guidelines.

Figure 5. Enlarge view of Angular diffuser

Figure 9. Velocity distribution graph for flat diffuser

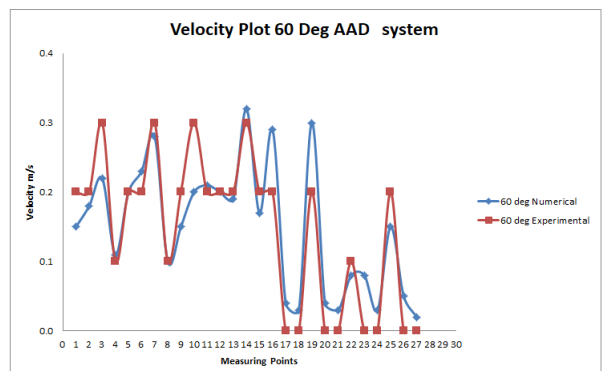**Figure 6.** Velocity Plot for OT with a flat diffuser

Figure 10. Velocity distribution graph for AAD system with 60° angle
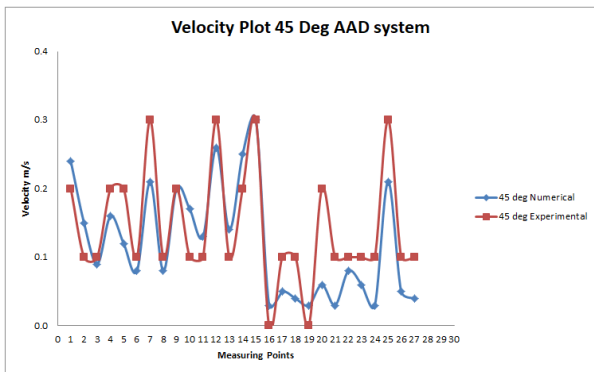
Figure 11. Velocity distribution graph for AAD system with 45º angle

From Figure 12 it is observed that in case of flat diffuser air velocity in non critical zone vary from 0 to 0.4 m/s, which results in turbulence formation in that area, but in other case i.e. 60º and 45º AAD system the velocity variation in non critical zone is less which reduced the turbulence formation. For visualization of air flow pattern in AAD system with different angle 90, 60, 45,various images have been taken by the camera and the smoke is visualized inside the OT as shown in Figures 13-14. The smoke flow spreads and it forms vortices when it touches the operating table and surgeons and its flows in the opposite direction of the primary flow. Figures 13 and 14 present the path line exit from AAD system with 60˚ and 45˚ angle of louvers. The smoke flow from diffuser reaches to the middle plane of OT with minimum turbulence. The throw of AAD system seems as long and equally spread in whole OT; therefore there is no vortex formation in OT.
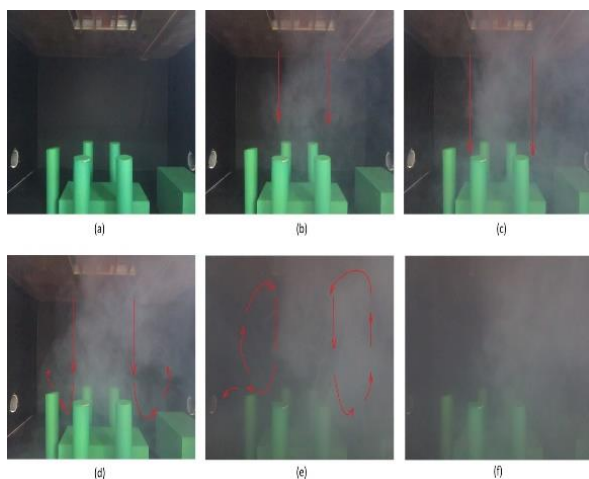

Figure 12. Smoke visualization for flat diffuser (a) No smoke (b) Initial stage (c) Straight flow (d) vortex formation (e) mixing of primary and secondary flows (f) high amount of vortex formation and poor exhaust
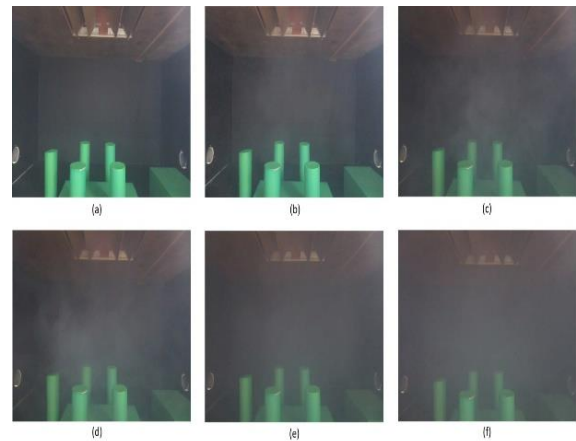

Figure 13. Smoke visualization for AAD system with a 60º angle (a) No smoke (b) Initial stage (c) Angular flow (d) Angular spread of smoke (e) Spreading of smoke beyond critical zone (f) equally spread smoke with a minimum vortex in whole OT with a proper exhaust.

Figure 14. Smoke visualization for AAD system with 45º angle (a) No smoke (b) Initial stage (c) Angular flow (d) Angular spread of smoke (e) Spreading of smoke beyond critical zone (f) equally spread smoke with minimum vortex in whole OT with proper exhaust
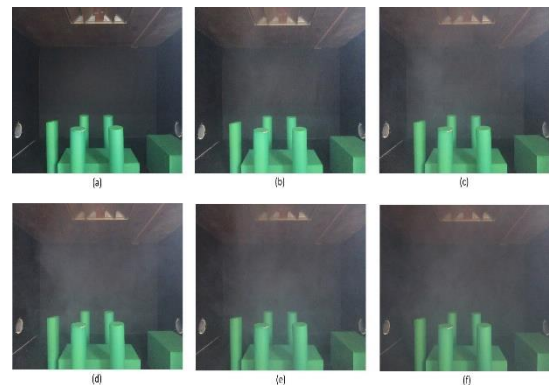
4. CONCLUSION

The complexity in the HVAC system in hospital health care premises is increased as an increase in the use of modern technology and instrument.The presented work aimed to study and visualize the formation of turbulence of airflow inside an OT with a traditional flat diffuser with velocity distribution in OT. Air flow pattern inside scale model of OT with different inlet angleat constant velocity 0.4m/s is visualized numerically by streamlines and experimentally by the smoke test. The flat diffuser provides uniform airflow at the critical zone only but badly increases the reverse flow and turbulence in critical zone when the obstruction arrives. This effect may cause bacterial dispersion throughout the room and temperature draft in OT. The remedy over such problem is to increase

the inlet surface area of the flat diffuser or to provide angular flow over the critical zone. It is achieved by implementation of AAD system. AAD ventilation system provides unidirectional flow in whole OT with maximum sterile zone.It will not affect the interior of OT, it cover the required air flow volume of OT with unidirectional flow as per requirement. It has easy fabrication and installation process.

5. ACKNOWLEDGMENTS

6. REFERENCES

1. Ho, S.H., Rosario, L. and Rahman, M.M., "Thermal comfort enhancement by using a ceiling fan", *Applied Thermal Engineering*, Vol. 29, No. 8-9, (2009), 1648-1656.

2. Van Gaever, R., Jacobs, V.A., Diltoer, M., Peeters, L. and Vanlanduit, S., "Thermal comfort of the surgical staff in the operating room", *Building and Environment*, Vol. 81, No., (2014), 37-41.

3. Jones, W.P., "Air conditioning engineering, Routledge, (2007).

4. Paul Ninomura, P. and Richard Hermans, P., "Ventilation standard for health care facilities", *ASHRAE Journal,* Vol. 50, No. 10, (2008), 52 - 57.

5. Health, A.A.o.A.f. and Institute, F.G., "Guidelines for design and construction of hospital and health care facilities, Aia Press, (2001).

6. Normung, D.I.f., *D*in 1946-4, ventilation and air conditioning—part 4: Vac systems in buildings and rooms used in the health care sector. 2008, DIN Berlin, Germany.

7. Humphreys, H. and Taylor, E., "Operating theatre ventilation standards and the risk of postoperative infection", *Journal of Hospital Infection*, Vol. 50, No. 2, (2002), 85-90.

8. Balaras, C.A., Dascalaki, E. and Gaglia, A., "Hvac and indoor thermal conditions in hospital operating rooms", *Energy and Buildings*, Vol. 39, No. 4, (2007), 454-470.

9. Chow, T.-T. and Yang, X.-Y., "Performance of ventilation system in a non-standard operating room", *Building and Environment*, Vol. 38, No. 12, (2003), 1401-1411.

10. Chow, T.-T. and Yang, X.-Y., "Ventilation performance in operating theatres against airborne infection: Review of research activities and practical guidance", *Journal of Hospital Infection*, Vol. 56, No. 2, (2004), 85-92.

11. Chow, T. and Yang, X., "Ventilation performance in the operating theatre against airborne infection: Numerical study on an ultra-clean system", *Journal of Hospital Infection*, Vol. 59, No. 2, (2005), 138-147.

12. Sadrizadeh, S., Holmberg, S. and Tammelin, A., "A numerical investigation of vertical and horizontal laminar airflow ventilation in an operating room", *Building and Environment*, Vol. 82, No., (2014), 517-525.

13. Liu, J., Wang, H. and Wen, W., "Numerical simulation on a horizontal airflow for airborne particles control in hospital operating room", *Building and Environment*, Vol. 44, No. 11, (2009), 2284-2289.

14. Ufat, H., Kaynakli, O., Yamankaradeniz, N. and Yamankaradeniz, R., "Investigation of the number of particles in an operating room at different ambient temperatures and inlet velocities", *International Journal of Ventilation*, Vol. 17, No. 3, (2018), 209-223.

15. Ayremlouzadeh, H. and Ghafouri, J., "Computational fluid dynamics simulation and experimental validation of hydraulic performance of a vertical suspended api pump", *International Journal of Engineering-Transactions B: Applications*, Vol. 29, No. 11, (2016), 1612-1620.

Design of Air Distribution System for Operation Theatre Using Flow Visualization Techniques to Improve Flow Characteristics

S. D. Rahate, A. D. Sarode

Department of Mechanical Engineering, Lokmanya Tilak College of Engineering, University of Mumbai, Navi-Mumbai, India

P A P E R I N F O	چکیده
	تاتر عملیاتی (OT) مهمترین منطقه ای است که کنترل دقیق آن بر کیفیت هوای داخل خانه لازم است زیرا به طور مستقیم با سلامت بیمار و دوره بهبود آن مرتبط است. این کار با هدف بررسی و تجسم توزیع جریان هوا از سیستم تهویه هوا پراکنده هوا معمولی و سیستم تهویه هوا زاویه ای تازه طراحی شده در OT انجام شده است. سیستم توزیع هوای زاویه ای (AAD) به گونه ای طراحی شده است که ، هوای مطبوع با زاویه ای در OT پرتاب می شود تا بتواند حداکثر مساحت OT را با جریان چند لایه و حداقل محل نصب پوشش دهد. عملکرد سیستم توزیع هوای زاویه ای (AAD) با زاویه متفاوت انتشار پریز ، سرعت ورودی ٤.٠ میلی متر در ثانیه و الگوی جریان هوا با کمک شبیه سازی عددی و راه اندازی آزمایشی مدل نمونه OT بررسی شده است. بررسی تجربی جزئیات بر روی نمونه اولیه با کمک تست دود و روش هدف دوربین (CTM) انجام می شود. هر دو نتایج عددی و تجربی یک الگوی مشابه از توزیع هوا برای تنظیمات مختلف طراحی نشان داد. *doi: 10.5829/ije.2020.33.01a.19*