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

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. Therefore 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 18-22°C 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.rahate36@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.

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 m² surface area) and Four exhausts (0.00580 m² 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:

\[ \nabla \cdot (\rho \vec{v}) = S_m \]  

Momentum:

\[ \nabla \cdot (\rho \vec{v} \vec{v}) = -\nabla p + \nabla \cdot (\vec{F}) + \rho \vec{g} \]  

\[ \vec{r} = \mu \left[ (\nabla \vec{v} + \nabla \vec{v}^T) - \frac{2}{3} \nabla \cdot \vec{v} \right] \]  

Energy conservation:

\[ \nabla \cdot (\rho \vec{v} + p \rho \vec{v}) = -\nabla \cdot \left[ \rho \nabla \nabla \nabla \right] + S_h. \]  

Turbulence Kinetic Energy (k):

\[ \frac{\partial}{\partial t} (\rho k u_i) = \frac{\partial}{\partial t} \left[ \left( \mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right] + G_k + G_b - \rho \varepsilon - Y_{\theta} \]  

Dissipation Rate of turbulence kinetic energy (ε):

\[ \frac{\partial}{\partial t} \left( \rho \varepsilon u_i \right) = \frac{\partial}{\partial t} \left[ \left( \mu + \frac{\mu_t}{\sigma_\varepsilon} \right) \frac{\partial \varepsilon}{\partial x_j} \right] C_{1\varepsilon} \frac{\varepsilon}{k} \left( C_k + G_b \right) - C_{2\varepsilon} \frac{\varepsilon^2}{k} + S_{\varepsilon} \]

\[ C_{1\varepsilon}, C_{2\varepsilon}, \sigma_k, \sigma_\varepsilon \] are constant of turbulence model and their values are \( C_{1\varepsilon} = 1.44, C_{2\varepsilon} = 1.92, \sigma_k = 1.0, \sigma_\varepsilon = 1.3 \)

\( \nabla \) is partially derivatives of quantity in all direction \( G_k, G_b \) is generation of turbulence kinetic energy due to the mean velocity gradient, \( S_h, S_b \) is strain tensor with respect heat dissipation, \( S_b, S_b \) is user defined source term obtained by \( \kappa \) and \( \varepsilon \) model equation, \( S_b \) 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

\[ \begin{array}{|c|c|c|} 
\hline
\text{Material} & \rho (\text{kg/m}^3) & C_p (\text{J/kg k}) & K (\text{w/mk}) \\
\hline
\text{Wood} & 700 & 2310 & 0.173 \\
\text{Air} & 1.225 & 1006 & 0.02435 \\
\hline
\end{array} \]
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 586479 as 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 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: - ± (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 anemometre 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 edge 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: - 900 watt, heat up time - 12 min, capacity-3 liters) with fog fluid (white cloud) is used.
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 acceptable range as mentioned in national and international standards and guidelines.
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. 60° and 45° 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.

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 angle at constant velocity 0.4 m/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
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

5. ACKNOWLEDGMENTS

The authors acknowledged the financial support from University of Mumbai under Minor Research Grant Project (Research Project Number 384) in the academic year 2018-2019 in Lokmanya Tilak College of Engineering, Navi Mumbai, India

6. REFERENCES

1. Ho, S.H., Rosario, L. and Rahman, M.M., "Thermal comfort enhancement by using 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 performance of a vertical suspended api pump", Applied Thermal Engineering, Vol. 29, No. 11, (2009), 138-147.
5. 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.
6. 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.
7. Ufar, 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. 3, No. 3, (2018), 209-223.
8. Ayremlouzadeh, H. and Ghafoori, 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