An experimental and numerical study of air flow pattern and temperature distribution of angular air distribution system in hospital operation theatre

Swati Rahate\textsuperscript{1,a}, Avinash Sarode\textsuperscript{1,b}

\textsuperscript{1}Mechanical Department, Lokmanya Tilak College of Engineering, Navi-Mumbai.

\textsuperscript{a}Swati.rahate363@gmail.com, \textsuperscript{b}avinashsarode@gmail.com

Abstract: - It is necessary to characterize the air flow pattern and temperature distribution in Operation Theatre (OT) to optimize the indoor air quality to enhance the high performance of hospital. Mostly Laminar air flow system is used in OT, which covered the surgical zone. The temperature draft observed between surgical and nonsurgical area which result in uncomfortable environment to patient and surgeon. To overcome such problem Angular Air Distribution (AAD) system is designed which covered the both area of OT. The performance of AAD system with varying inlet diffuser angle at constant inlet velocity on Scale model of OT is investigated with the help of CFD software and compared with experimental setup. Detail experimental investigation is done on a scale model of OT with the help of a smoke test and camera target method (CTM). Both numerical and experimental results showed a similar pattern of air and temperature distribution for the various design configurations.

1. Introduction
As ventilation system play important role in HVAC to keep OT environment free from bacteria. The poorly ventilated system not only affects the health of patient and surgeon from infection but also feel them uncomfortable [1]. Many national and international standards and guidelines are available for every country to design a ventilation system with environmental parameters for OT. LAF system provides unidirectional ultra clean air flow to protect the surgical zone only. If the whole OT is designed for LAF system then installation area and air circulation rate (ACH) of the ventilation system is increased which results in an increase in energy consumption and maintenance of system [2]. The traditional ventilation systems of OT provide the conditioned air to the critical zone only by maintaining laminar flow. Hospitals and other health care facilities are complex environments that require comfort and control conditioned air with flow pattern [3]. The persons present in OT such as patient and surgeon suffer from surgical site infection (SSI) and postoperative infection [4]. Indoor obstacles such as personnel, operating lamp, surgical instrument and movement of the surgeon can easily affect the unidirectional flow pattern of the ventilation system. Alternate methods are mixing, displacement, Vertical and horizontal ventilation system, which differs by the position of supply and an exhaust port in OT [2]. 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 way that it will distribute the air in laminar pattern in whole OT.
Many experimental and numerical analyses have been presented about ventilation system and thermal comfort in ORs. Balaras, C.A et al. [5] had presented an overview of the general indoor condition and air quality in OTs. They 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 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 (90fpm), there is two types of LAF one is horizontal and another is vertical. In this literature velocity is measured by using hot wire anemometer and compared it with a mathematical k-e model for validation purpose [6, 7, 8]. Zang Rui et al. [9] explained that by improving airflow pattern, the particle deposition in the critical zone can be reduced but its effect becomes reduced if we increase the air change rate. He concluded that 0.25 m/s is the best velocity. Villafruela et al. [10] analyzed the effective design of the ventilation system for isolation rooms which reduce the risk of spreading infections of infected patients. Liu et al [11] investigated an alternative of horizontal flow pattern and air flow performance in an OT with a dimension of 300cm long, 296 cm wide and 240 cm high. He also evaluates the effectiveness of the horizontal unidirectional airflow to control infectious airborne particles in operation theatre. He also stated that downward unidirectional air flow has some disadvantages.

In view of above contradictory results and worldwide increases in health care costs and increasing difficulties in financing and providing all modern medical advances in OT Angular Air Distribution (AAD) System is designed whose primary goal is to provide unidirectional flow in whole OT and reduced the temperature draft to provide comfortable environment to patient and surgical staff. To test the designed AAD system, prototype scaled model of OT with 1:4 ratio is fabricated. Initially the scale model of OT is examined by numerical simulation by varying inlet angle, inlet velocity and temperature fluent software is used. The boundary conditions of numerical results are tested with grid independency test. The numerical results are validated with the help of experimental readings of scale model of OT The air flow pattern of AAD system in OT with various inlet angles are tested with the help of smoke test. The flow pattern obtained by smoke of AAD system is capture by Camera Target Method (CTM) and it is validated with CFD results. To check the human comfort and effectiveness of air distribution system, the obtained result is compared with the international standard parameter for ventilation system such as Air Diffusion Performance Index (ADPI).

2. Methodology

2.1 Geometrical similarity criteria

A reduced scale model is used to conduct a series of model experiments and investigate of ventilation system of OT. For geometrical similarities between scale model and prototype model, scale model is made with 1:4 ratios. The geometrical similarity is consider as follows

$$\frac{L_p}{L_m} = \frac{W_p}{W_m} = \frac{H_p}{H_m} = L_r \quad (1)$$

$$\frac{A_p}{A_m} = L_r^2 \quad (2)$$

$$\frac{V_p}{V_m} = L_r^3 \quad (3)$$

Where $L_p, W_p, H_p$ is length, width and height of prototype model. $L_m, W_m, H_m$ is length, width and height of Scale model. $L_r$ Is Geometrical ratio as explain in equation (1), (2) and (3). Table 1 shows the ratio of geometrical similarity between scale and prototype model. From table 1 $L_r = 4$.

| Geometrical Similarity | Actual (Prototype) Model | Ratio | Scaled Model |
|------------------------|--------------------------|-------|--------------|
| Length (m)             | 6                        | 4     | 1.5          |
| Width (m)              | 6                        | 4     | 1.5          |
| Height (m)             | 3                        |       | 0.75         |
| Area ($m^2$)           | 36                       |       | 2.25         |
| Volume ($m^3$)         | 108                      |       | 1.6875       |
2.2 CAD modeling

A 3D model of OT room geometry that outline by rectangular shaped room of wooden material with 1.524 m (L) x 1.524 m (B) x 0.75 m (H) dimension as shown in figure 1. The room is equipped with 5 people and one operating bed, anaesthesia machine, surgical lamp and operating bed, which is located at middle of the OT under the area of Laminar flow. The 5 surgical staff is standing around the operating table. They were modelled in cylindrical shaped with 0.0762m diameter and 0.3048m height. Anaesthesia machine of size 0.075 m (L) X 0.0625 m (W) X 0.425 m (H) is placed head side of patient. The supply ventilation system is installed in such way that its flow will cover the critical zone or flow will wash out the critical zone. Square ceiling supply diffuser (0.5 m (l) X 0.5 m (w)) with 4 number of louvers equally spaced from edged of diffuser is located in central zone of ceiling, that guide the unidirectional and angular flow by varying angle of louvers as shown in figure 1. Four exhausts (0.00580 m2 cross section area of each) are arranged over two of opposite side wall and allow the internal air to outflow.

![Figure 1](image)

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

Four different additional configurations of AAD system as shown in figure 1 are to be study starting from flat to angular diffuser. For these four different ventilation systems constant air flow rate 25 ACH are assumed.

2.3 Numerical Solution

The numerical solution is carried out in commercial software which allowing solution of the governing equation by finite element approach such as CFD (Fluent). Fluent is a general purpose commercial CFD software package that solves the Navier–Stokes equations using a finite volume method and SIMPLEC algorithm. In this work, Ansys-Fluent 15.0 was used to solve the momentum, continuity and turbulence equations for fluid flow. A solid and fine tetrahedron elemental auto-mesh is used to generate mesh model. The software ANSYS mesh was used for meshing of OT geometry with many elements. The different mesh size is considered for mesh independence study. Tetrahedron cells were used with varying number of elements of 256916, 287258, 312761, 39844, and 586479. The velocity plot for different mesh size is plotted and it was approximately same for last four numbers of elements.
The criteria for converged numerical solution are taken as 1e-3 RMS with element size 586479 as shown in figure 2.

Airflow pattern was determined by solving a set of partial differential equations that consist of mass, momentum and energy conservations, written in equation (4), (5), and (6), below respectively:

\[ \nabla \left( \rho \vec{V} \right) = \mathbf{S}_m. \]  
\[ \nabla \left( \rho \vec{V} \vec{V} \right) = \nabla p + \nabla (\mathbf{r}) + \rho \mathbf{g} \]  
\[ \nabla \left( \vec{V} \left( \rho \vec{V} + p \right) \right) = -\nabla \left[ \sum \mathbf{h}_i \mathbf{J}_i \right] + \mathbf{S}_h. \]  

This airflow study used the standard k–\( \varepsilon \) turbulence model equation (7) and (8) that consisted of the following Navier-Stokes equations:

**Turbulence Kinetic Energy (k):**

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

**Dissipation Rate of turbulence kinetic energy (\( \varepsilon \)):**

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

Where \( G_k = -\vec{u}_i \vec{u}_j \frac{\partial u_j}{\partial x_i} \) to evaluate this parameter Boussinesq hypothesis is used \( G_k = \mu_t S^2 \) where the modulus of the mean rate of strain tensor is defined as \( S \equiv \sqrt{2} S_{ij} S_{ij} \). \( C_{1e} = 0.09, C_{2e} = 1.44, \sigma_k = 1.0, \sigma_\varepsilon = 1.3. \)

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 are considered as standard wall function including interior of OT. The values of physical properties of material used in numerical simulation are listed in table 2.

**Table 2. Physical Properties of material used in numerical simulation**

| Material | \( \rho \) (kgm\(^{-3}\)) | \( C_p \) (j/kg\(^{o}k\)) | K (w/m\(^{o}k\)) |
|----------|----------------|----------------|----------------|
| Wood     | 700            | 2310           | 0.173          |
| Air      | 1.225          | 1006           | 0.02435        |
2.4 Experimental setup
The general layout of experimental setup is shown in figure 3. It consists of three main parts such as air distribution system i.e. supply diffuser, Air Conditioning unit or Air handling unit (AHU) and testing room. Air handling unit is simple unit of vapour compression refrigeration cycle. Fan is pushing the atmospheric air over evaporating coil to get conditioned cooled air. The OT scale room model made of wooden material. This is scale by 1:4 ratio of full scale of prototype. OT room is fabricated by the plywood of 12 mm thick for three side and floor and top surface, front side of testing room is made by acrylic sheet for capturing the images of smoke test. The round supply duct of 10 cm diameter is used to supply the conditioned air from AHU outlet to inlet of test room. Four exhausts are connected to exhaust fan with the help of exhaust duct.

![Figure 3. General layout of experimental setup](image)

2.5 Measurement procedure and instrumentation
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 the help of camera target method. 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 generates smoke which is accumulated in the mixing box. It is connected to inlet diffuser of OT by means of flexible ducts. 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. Four outlets are used which is connected to exhaust through exhaust manifolds. 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 litres) with fog fluid (white cloud) is used. The blower is inbuilt in the smoke generator. The interior features of OT and the white smoke is targeted and its images are captured by high definition digital camera. Since the camera is fixed at its position, the multiple proportionate images are captured and compared on the bases of smoke distribution inside the OT. 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.
As the human comfort is depend on the temperature draft in occupied zone. To investigate the temperature draft in whole OT temperature measurement at various points is needed. According to temperature reading the effectiveness of air distribution system is calculated. For temperature measurement fixture is design and fabricated as shown in figure 4.

As the geometry of scaled OT room is symmetrical with respective central plane, the temperature measurement is taken at one side of central plane of OT. For temperature measurement at different location, the temperature measuring fixture is placed at different location. To measure temperature draft in critical zone and beyond that zone total nine line of temperature measuring fixture is used as shown in figure 4. The temperature is measured at different plane 700 mm, 500 mm and 200 mm from the floor. The 700 mm height plane is near to ceiling, 500 mm plane at middle of OT, and 200 mm height plane near to operating bed.

All velocity measurement is carried out by temperature compensate Hot wire anemometer. Hot wire anemometer probe is connected to the display and storage unit. Velocity measurement was performed inside the OT, Diffuser inlet and AHU outlet. The length of probe is 3 meter long to measure the velocity at middle plane of OT. The velocity is measured at the same point of temperature sensor. To access the all 27 point of temperature sensor inside the OT, side wall of scale model is drilled with the hole to insert the probe of anemometer at different point for velocity measurement. The coordinate of velocity measuring point in OT with line fixture and its location are shown in figure 5. Initially velocity is measured at AHU outlet, then at diffuser inlet and outlet, at exhaust outlet of OT.
The widely accepted and design index that quantifies the performance of air distribution system when considering the spatial uniformity of air distribution to thermal comforts is air diffusion performance index (ADPI). Miller introduced the ADPI concept in HVAC industries for research by designing different types of air distribution system. ADPI is defined as the percentage of occupied zone falling into the acceptable velocity and temperature region determined by measuring local Effective Draft Temperature (EDT). ADPI is based on air speed and effective draft temperature (EDT) as mentioned in equation (9). ADPI method required to calculate EDT.

\[
EDT = (T_P - T_R) - 8 \times (V_P - 0.15)
\]  

(9)  

Where \( T_P \) is air temperature at a test point in °k, \( T_R \) is the average temperature of room or operation theatre and \( V_P \) is local air velocity in m/s. For comfort, the EDT should be within −1.7°C to +1.1°C and the air velocity should be less than 0.36 m/s. The objective of air distribution system design is to select and place the supply air diffusers in such a way that the ADPI approaches 100 %. The ADPI provides a rational way of selecting air diffusers. The ADPI as mentioned in equation (10) is defined as the percentage of measurements taken at many locations in the occupied zone of space that meets EDT criteria of −1.7°C to +1.1°C, that is

\[
ADPI = \left( \frac{N_\theta}{N} \right) \times 100
\]  

(10)  

Where \( N \) is the total number of locations at which observations have been made, and \( N_\theta \) is the number of locations at which the effective draft temperature is within −1.7°C to +1.1°C.

3. Result and discussion

3.1 Air flow pattern

CFD analysis is performed on the featured scaled geometry by using Ansys Fluent. The result of flat diffuser for air flow pattern, and turbulence plot is as shown in figure 6.

![Image of air flow pattern and turbulence plot](image_url)
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 vortices cause temperature draft (conada effect) and turbulence in the flow of harmful particles along with the airstream flow over the surgeons and remain in the critical zone. Turbulence kinetic energy remains low in the region from inlet diffuser to the critical zone but it increases in the obstacles offered by the surgeons. Also, the velocity is high in the same region. Hence, the primary flow carries bacteria and mixes with fresh incoming airflow.

Figure 7. Distribution of air stream, turbulence energy and smoke visualization in OT with 30° louvers angle

Figure 8. Distribution of air stream, turbulence energy and smoke visualization in OT with 45° louvers angle

Figure 9. Distribution of air stream, turbulence energy and smoke visualization in OT with 60° louvers angle

Figure 7, figure 8 and figure 9 presents the path line exit from AAD system with 30°, 45° and 60° angle of louvers. The smoke flow from diffuser reaches to the middle of OT with minimum turbulence. The throw of AAD system seems as long and equally spread in whole OT; therefore, there is minimum vortex formation in OT.
3.2 Velocity and temperature measurement

The temperature on all measuring point i.e. 15 points inside the critical zone area is in range of 25±/−2°C in all diffuser. But in case of flat diffuser outside the critical zone the temperature is higher than critical zone. Which feel the drastic temperature draft to surgeon and patient which affect the human comforts. But in case of AAD system this temperature draft is less as compared to flat diffuser system. At the same time, time required to achieve equilibrium temperature with help of AAD system in OT is less as compared to flat diffuser system which further result in reduction in energy consumption. The temperature range and its distribution in OT with flat and AAD system is shown in figure 10 and figure 11.

![Figure 10. Distribution of Temperature in OT with 90° and 30° louvers angle.](image1)

![Figure 11. Distribution of Temperature in OT with 45° and 60° louvers angle.](image2)

From Figure 12 and Figure 13, 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 20 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.
Figure 12. Distribution of velocity in OT with 90° and 30° louvers angle

Figure 13. Distribution of velocity in OT with 45° and 60° louvers angle.

3.3 Air Diffusion Performance Index (ADPI)

ADPI model consider thermal distribution as well as the range of effective draft temperature and maximum local air speed in OT. Furthermore this work generates ADPI for all case (Case 1 -90°, Case 2 -30°, Case 3 -45°, Case 4 -60° ) of designed air distribution in critical zone and non-critical zone. The measurement shows that AAD system with 45° louvers angle generate ADPI more and equal to 80% in critical and non-critical zone as compare to other cases as shown in Figure 14, which results in
uniform temperature distribution with minimum temperature draft to get thermal comfort for occupant in OT.

4. Conclusion
Angular Air Distribution system can provide an alternate ventilation system for operation theatre. The results of temperature and velocity distribution in OT with AAD system is compared with Laminar air distribution system. The causes of uneven distribution of flow in laminar air distribution system are visualized by both theoretical and experimental method. The flat diffuser or laminar air distribution system provides uniform airflow at the critical zone only but badly increases the reverse flow and turbulence in critical zone when the obstruction arrives. The tendency of mixing of primary and secondary flows is higher therefore the removal of used air throughout the room is not effective. 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 so that the removal of used air can be distributed evenly and exhausted quickly. It is achieved by implementation of AAD system by setting 45° louver angle which guides the incoming air in angle and distributes uniformly throughout the OT. It is also observed that AAD system with 45° louver angle satisfied the ADPI standard for air distribution system in critical as well as in non-critical zone which results in minimum temperature draft. The presented work methodology sets a good agreement between the numerical and experimental techniques and results.

References
[1] Ho SH, Rosario L, Rahman MM. Three-dimensional analysis for hospital operating room thermal comfort and contaminant removal. Applied Thermal Engineering. 2009 Jul 1;29(10):2080-92.
[2] Sadrizadeh S, Holmberg S, Tammelin A. A numerical investigation of vertical and horizontal laminar airflow ventilation in an operating room. Building and Environment. 2014 Dec 1;82:517-25.
[3] Woods JE, Braymen DT, Rasmussen RW, Reynolds PE, Montag GM. Ventilation requirements in hospital operating rooms—Part I: Control of airborne particles. ASHRAE transactions. 1986;92(2):396-426.
[4] Friberg B. Ultraclean laminar airflow ORs. AORN journal. 1998 Apr 1;67(4):841-51.
[5] Balaras CA, Dascalaki E, Gaglia A. HVAC and indoor thermal conditions in hospital operating rooms. Energy and Buildings. 2007 Apr 1;39(4):454-70.
[6] Chow TT, Yang XY. Performance of ventilation system in a non-standard operating room. Building and environment. 2003 Dec 1;38(12):1401-11.
[7] Chow TT, Yang XY. Ventilation performance in operating theatres against airborne infection: review of research activities and practical guidance. Journal of Hospital Infection. 2004 Feb 1;56(2):85-92.
[8] Chow TT, Yang XY. Ventilation performance in the operating theatre against airborne infection: numerical study on an ultra-clean system. Journal of Hospital Infection. 2005 Feb 1;59(2):138-47.
[9] Rui Z, Guangbei T, Jihong L. Study on biological contaminant control strategies under different ventilation models in hospital operating room. Building and environment. 2008 May 1;43(5):793-803.
[10] Villafruela JM, Castro F, San José JF, Saint-Martin J. Comparison of air change efficiency, contaminant removal effectiveness and infection risk as IAQ indices in isolation rooms. Energy and Buildings. 2013 Feb 1;57:210-9.
[11] Liu J, Wang H, Wen W. Numerical simulation on a horizontal airflow for airborne particles control in hospital operating room. Building and environment. 2009 Nov 1;44(11):2284-9.