PAPER • OPEN ACCESS

Numerical Investigation of Flow Field Near the Exhaust Opening Indoor

View the article online for updates and enhancements.

Numerical Investigation of Flow Field Near the Exhaust Opening Indoor

Chanjun Meng[1], Bo Yuan[1,2,3,*], Chuan He[1], Chunfu Xi[1], Liangming Pan[1,3]

[1]Key Laboratory of Low-grade Energy Utilization Technologies and Systems, Chongqing University, Chongqing, 400044, China

[2]State Key Laboratory of Coal Mine Disaster Dynamics and Control, Chongqing University, Chongqing, 400044, China

[3]Postdoctoral Research Station of Power Engineering & Engineering Thermophysics, Chongqing University, Chongqing, 400044, China

*Corresponding author: boyuanyuan@cqu.edu.cn

Abstract. Airflow field and effective region for pollution discharge were predicted around a circular flanged exhaust opening, three different turbulent models were employed to simulate airflow field near the exhaust opening indoor. Numerical results show that air velocity decreases gradually in radial direction and axis direction away from origin, and changes most dramatically at corner of the opening. The projection of velocity contour is similar to a semi-ellipse, but when the air velocity decreases, it gradually changes the hemisphere with larger outer radius. When the velocity is 2 m/s, the long axis of the semi-ellipse is 0.70 times the diameter of the exhaust port and the short axis is 0.45 times the diameter of the exhaust port. When the velocity is 0.3m/s, the diameter of the outlet is 1.60 times of that of the long axis and 1.55 times of that of the short axis. The effective area of pollutant emission is similar to the equivalent surface with velocity of 0.3m/s, and its shape is semi-elliptic.

1.Introduction

The best way to ensure good industrial hygiene is to adopt indoor air conditioning and pollutant control by suction inlet equipment [1-4]. Air ventilation with exhauster makes a key influence on the circulation of the air indoor, in which many researchers have taken a lot of investigations in this field. The results show that in the internal space of the room, the air exhaust device generates suction effect, so that the air near the suction device is affected, and generated a directed suction speed from static state to the suction surface.

Far away from the exhaust opening, air speed is too low to eliminate pollutants effectively, so velocity magnitude field is necessary to investigate. Some researchers obtained a correlation for velocity on the centerline of exhaust opening[5-6]. A research on the velocity magnitude field was made and came to a conclusion that velocity magnitude contours and streamline are similar in geometrically similar situation[7]. In 1952, the similarity principle of Dalla Valle [8] has been perfected, and the similarity principle was perfectly applied to the circular suction vent. In 1970, Drkal [9] came up with an analytic solution for flanged exhaust based on potential flow theory and acquired component velocity of the flow field around a circular export. Flynn and Ellenbecker [10] carried out empirical correction for the model. In 1996, F. Cascetta and Laura. Bellia [11] analyzed and compared the velocity values on the axial line with the empirical correlation and experimental results, aimed at the

flow field near the circular inlet, at the same time the condition of the isokinetic line near the suction outlet obtained by the experiment. In 1993, Ilpo. Kulmala [12] conducted a numerical calculation on airflow fields of exhaust openings and verified the validity of numerical simulation applied in the analysis of flow field around it. In 2004, M. T. Shervani Tabar [13] analyzed airflow field of flanged exhaust openings with CFD technique and found a good agreement on flow centerline velocity and velocity magnitude contours between numerical simulations and potential flow solutions. However, little research reported about the region and velocity distribution around the exhaust opening for pollution discharge in the room.

In this paper, the effective region and air velocity distribution near the exhaust port were studied by numerical simulation. The diameter of the exhaust opening is 0.173 m, the distance between the center of the exhaust opening and the roof is A/2, the flow rate is 0.14 m^3/s, and the *Re* is 68120.

2. Simulation method

Considering incompressible fluid. Continuity equation and momentum equation are as follows:

$$\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x_i}(\rho u_i) = 0 \qquad (1)$$

$$\frac{\partial}{\partial x_j}(\rho u_i u_j) = -\frac{\partial p}{\partial x_i} + \frac{\partial \tau_{ij}}{\partial x_j} + \rho g_i \qquad (2)$$

$$\tau_{ij} = \mu\left[\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right) - \frac{2}{3}\mu\frac{\partial u_t}{\partial x_t}\delta_{ij}\right] \qquad (3)$$

In this paper, according to the results of experiment from Furio Cascetta [14]., *RNG k-ε* turbulence model and standard wall function were selected to simulate the model.
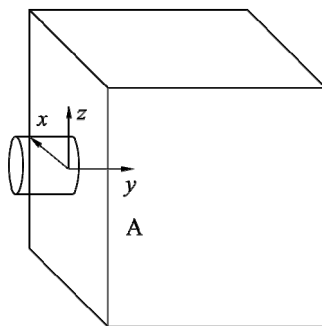

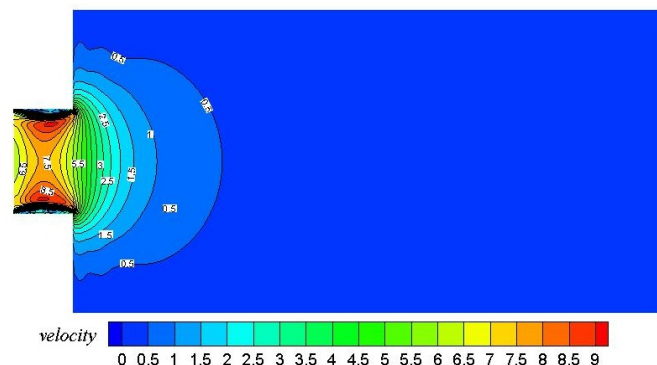Figure 1.The location diagram of the flanged circular exhaust opening.

Figure 2. Velocity magnitude contours in vertical and symmetry plane of opening, A/2=0.25.

For simplicity, the model is a single-wall model with an exhaust port. In the study, the length of the simulated domain is 1 m, the diameter of the opening is d=0.173 m, and the distance between the opening and the top a/2 varies within 0.25 m, 0.45 m and 0.5 m. Figure 2-4 shows the results of velocity level contours on the vertical and symmetrical planes of flange circular exhaust ports in the study.
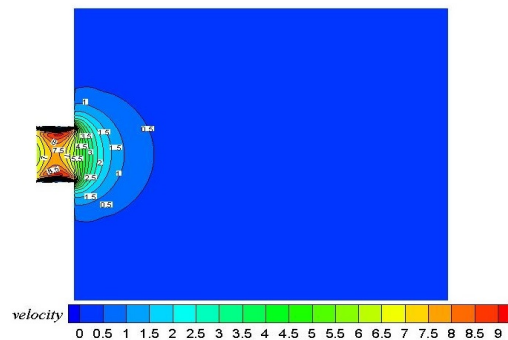
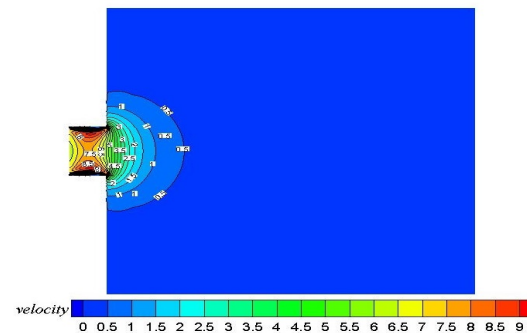Figure. 3 Velocity magnitude contours in vertical and symmetry plane of the opening.

Figure.4 Velocity magnitude contours in vertical and symmetry plane of opening, A/2=0.5.

When A/2 is 0.25 meters, the velocity trend is affected by the wall. When A/2 is 0.45 m and 0.5 m, the wind speed is zero at the boundary around the single wall, indicating that the influence area is smaller than the simulation area. Therefore, taking A/2 as 0.5m, and the ratio of the diameter of the hole to the distance between the hole center and the roof of the room as 0.35. A cubic simulation domain, 1m×1m×1m, is suitable when the ratio was set to be 0.17 in this numerical simulation.

3. Results and discussion

It considered that the shape of the exhaust opening is round, and the simulation area is symmetrical, Airflow field around the opening is also symmetric. In order to obtain the flow field and effective region, some analysis on the flow field of the vertical and symmetry plane of the flanged circular exhaust opening should be performed. Results of numerical simulation are fit for some similar situation with the application of similarity principle. Dimensionless velocity can be acquired by dividing velocity by the average velocity on cross-section that air flow though, which is shown as follows.

$$V^* = u/\overline{V} \tag{4}$$

Likewise, dimensionless axial distance in terms of y direction can be obtained.

$$y^* = y/D \tag{5}$$

Where y is distance of the origin in the axial direction, D is the opening diameter.
Dimensionless radial distance:

$$x^* = x/D \tag{6}$$

Where x is distance of the origin in the radial direction, D is the opening diameter.

In the influence of pressure gradient caused by the suction of exhauster upstream of which negative pressure exists in, air in region of influence flow toward exhaust opening. The Velocity magnitude contours show that the region of influence around circular flanged exhaust opening varies with the change of air speed. At high speed, the region can be described with a semi-ellipsoid, long axis is in the radial direction and major axis of the semi ellipsoid decreases in radial direction, however, minor axis in axial direction increases, as the air speed decreases. The two-dimension Velocity magnitude contours also show that different contours, the magnitude of which decrease as y^* increasing, are similar semi ellipse and become more and more similar with semicircle in an arrangement of layer after layer away from the origin. Through an observation on Figure 2, the direction of velocity of air in region of influence is toward the exhaust opening.

Research can be conducted through analyzing the maximum region influenced by suction in terms of removing pollutants in different average velocity of cross section. As figure 3 and 4 show. In order to exhaust the pollutant, speed of air have to be more than 0.3m/s. Shape of contours of different air speed induced by the suction of flanged exhaust opening is different. Two contours of air speed, 2m/s and 0.3m/s, were discussed respectively and after analyzing the vertical and symmetry plane of the flanged circular exhaust opening, there are two conclusions. When the air speed is 2m/s, the major

axis of the semi ellipsoid is 0.7 times the diameter of the circular opening, while the minor axis is 0.45 times the diameter. When the air speed is 0.3m/s, the major axis of the semi ellipsoid is 1.6 times the diameter, and minor axis is 1.55 times the diameter.
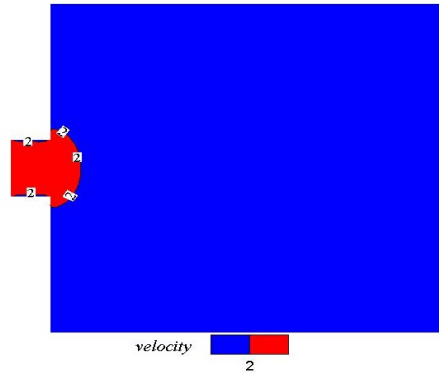

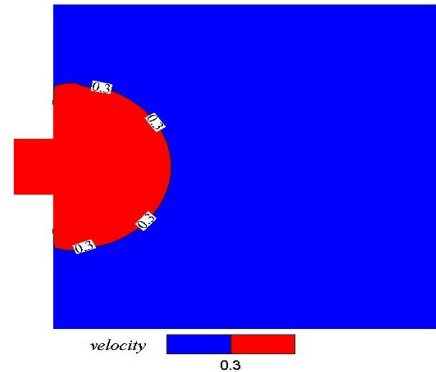Fig.5 Velocity magnitude contours in vertical and symmetry plane of the opening u=2m/s.

Fig.6 Velocity magnitude contours in vertical and symmetry plane of the opening, u=0.3m/s.

Analysis is performed on the dimensionless velocity in lines that are parallel with centerline and whose projection points extend away from the origin on y* axis, and in lines that are parallel with the circular opening plane and whose projection points extend away from the origin on x* axis to study the airflow field out of the centerline. Part of dimensionless projection points were shown in figure 7.
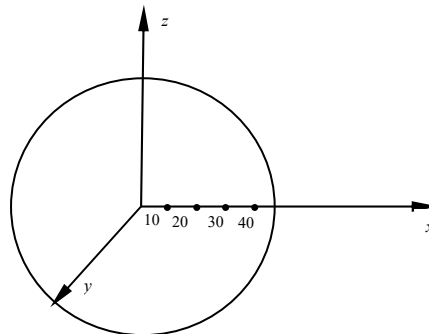

Fig.7 Dimensionless projection points of lines in parallel with the centerline in the opening plane.
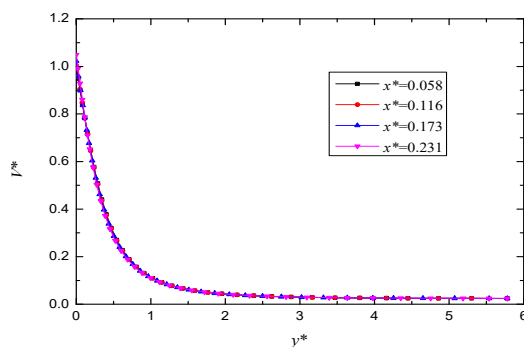

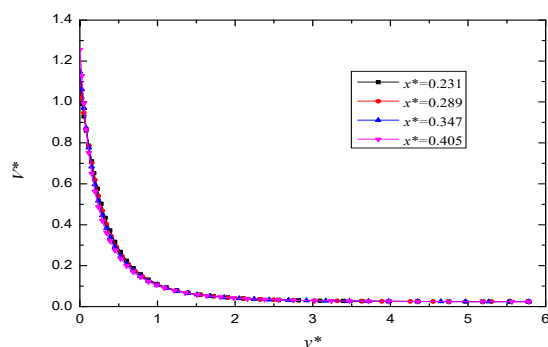Fig.8 Dimensionless velocity varying with $y*$ for different $x*$.

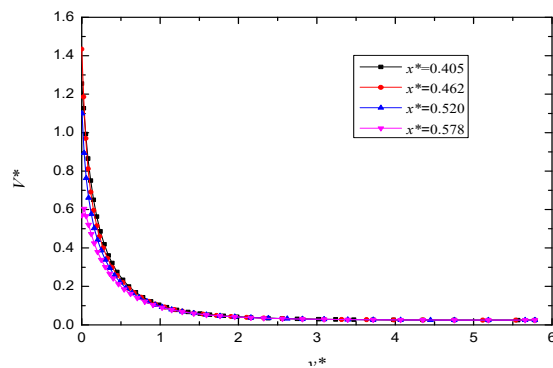Fig.9 Dimensionless velocity varying with $y*$ for different $x*$.

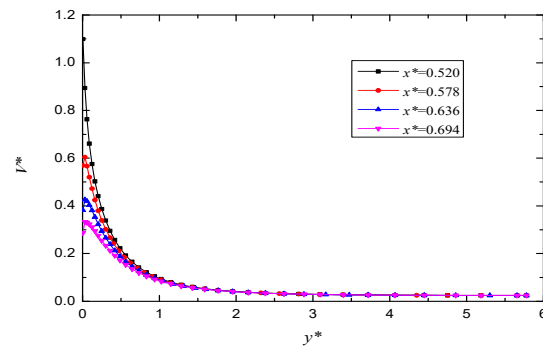Fig.10 Dimensionless velocity varying with y^* for different x^*.

Fig.11 Dimensionless velocity varying with y^* for different x^*.

In figure 8, the average cross-sectional velocity is decreased by more than 70% at $y^*=0.5$ for $x^*=0.231, 0.173, 0.116, 0.058$. The changes of V^* are gentle at $y^*>1$, and V^* is lower than 5% of the average cross-sectional velocity when $y^*>1.6$, which means air speed in those space is less than 0.3m/s and there is no suction effect for pollutants. In figure 9, likewise, velocity decreases rapidly, when $y^*<0.5$. Velocity is bigger away from the origin along x^* axis, when $y^*<0.1$. However, things are opposite, when $0.1<y^*<1$. Velocities in different x^* become consistent gradually when $y^*>1$ and all the v^* are less than 5%, no suction effect for pollutants existing, when $y^*>1.55$. In figure 10 and figure 11, the smaller x^* is, the bigger v^* is, at the same y^*, when $y^*<1$. While, V^* become consistent when $y^*>1$, too. That means airflow field changes more dramatically in interior zones.
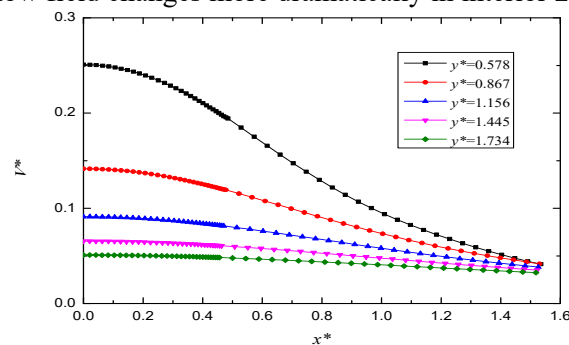

Fig.12 Dimensionless velocity varying with x^* for different y^*.

Figure 12 gives a view of dimensionless velocity varying with x^* for different y^*. V^* is less than 5% in the whole extent of x^* when $y^*=1.734$, which means there is no effect suction for pollutant, and the dimensionless variable $y^*=1.734$ is the ultimate distance in y direction for pollution discharge. All the V^* are more than 0.05 when $y^*<1.156$ and $0<x^*<1$, so there exists an effective region like a cylinder above the exhaust opening plane for pollutants elimination, called core effective region.

4. Conclusion
Within the present investigation, the main conclusions are listed in the following:

① *RNG k-ε* turbulence model and standard wall function are suitable for simulating the flow field around the circular flange exhaust opening in the room.

② The contour shapes are different under different air velocity. When the speed is 2m/s, the major axis of the semi ellipsoid is 0.7 times the diameter of the circular opening, while the minor axis is 0.45 times the diameter. When the air speed is 0.3m/s, the major axis of the semi ellipsoid is 1.6 times the diameter, and minor axis is 1.55 times the diameter.

③Numerical results by simulation with this model gave a profile of the effective region for pollution discharge, which is like a semi ellipsoid whose major axis is 1.6 times the diameter of the exhaust opening and minor axis is 1.55 times the diameter in a case that the average cross-section

velocity of the exhaust opening is 5.96m/s.

Acknowledgments

Reference

[1]   J L Alden and J.M. Kane. Design of Industrial Ventilation Systems [M], 5th edition, pp.20-27, Industial Press, New York (1982).

[2]   H. B. Awbi, Ventilation of Buildings. E&FN Spon (Chapman & Hall), London (1991).

[3]   D. B. Cameron, W. L. Johnston, etc. Air contaminant control investigation of a jet augmented local exhaust system. American Industrial Hygiene Association Journal 48, 56-61(1987).

[4]   ASHRAE, Handbook of Heating, Ventilating and Air-Conditioning Systems and Applications, American Society of Heating, Refrigerating and Air-Conditioning Engineers, New York (1987).

[5]   J.M.DallaValle, Exhaust Hoods, 2nd edition. Industrial Press, New York (1952).

[6]   B. Fletcher, Centerline velocity characteristics of rectangular unflanged hoods and slots under suction, Ann. Occup. Hyg. 20 (1977)141-146.

[7]   Dalla Valle, J. M., "Velocity Characteristics of Hoods Under Suction," Heating, Piping and Air Conditioning, 4:370-375 (1932).

[8]   J. M. Dalla Valle, Exhaust Hoods, Industrial. Press, New York, 2nd edn. (1952)

[9]   F. Drkal. Stromungsverhaltnisse bei runden Saugoffnungen mit Flansch [J].Z.Heiz,Luft.KLIM Haus,1970,21(6):271-273.

[10]   M. R. Flynn and M. J. Ellenbecker, The potential flow solution for air flow into a flanged circular hood. American Industrial Hygiene Association Journal 46(6), 318-322 (1985).

[11]   F. Cascetta. and Laura. Bellia. Velocity Fields in Proximity of Local Exhaust Hood Openings. Building and Environment. 31(5):451-459 (1996)

[12]   Ilpo Kulmala.Numerical calculation of air flow fields generated by exhaust openings.[J]. American Industrial Hygiene Association Journal, 1993, 37(5):451-467.

[13]   M. T. Shervani Tabar, M. H. Javareshkian, etc. Investigation of flow field around flanged hoods with Rectangular and Circular cross-sections[C]. Mechanical Engineering Conference, 2004:255-260.

[14]   F. Cascetta, Experimental Evaluation of the Velocity Fields for Local Exhaust Hoods with Circular and Rectangular Openings.[J].Building and Environment,1996,31(5):437-449.