Numerical Study of Blowing-Suction Ventilation Systems Performance

Pourya Shadkami Ahvazi\textsuperscript{a}, Navid P.Khabazi\textsuperscript{a}, Amirhossein Khayyaminejad\textsuperscript{a,*}

\textsuperscript{a} Department of mechanical engineering, Yazd university, Yazd, Iran

Abstract

In most industrial processes, toxic pollutants and vapors are produced and released, which cause various diseases in people working in industry and irreparable damage to the environment. Industrial ventilation systems are considered as one of the most effective methods of reducing and controlling gaseous pollutants and dust particles. One of the effective systems in industrial ventilation is blowing -suction ventilation systems. In this study, the effect of flow ratio (ratio of blowing flow to suction flow) and direction of blowing jet air on the performance of blowing-suction ventilation system was investigated numerically. The mixing parameter has been used as an indicator to measure the performance of the ventilation system. The results showed that the performance of the ventilation system blowing-sucking by reducing the current ratio improved exponentially. It was also found that one of the ways to improve the performance of the blowing – suction ventilation system is to reduce the direction of the blowing angle.

1. Introduction

The blowing – suction ventilation system is a very effective method for evacuating gaseous pollutants while having a stable performance and high resistance to external air flow [1-3]. Also, one of the advantages of this system is low energy consumption compared to other local ventilation systems [4]. The blowing -suction ventilation system consists of two general parts: Jet tail and suction hood. The tail section consists of one or more nozzles or an all-inclusive groove on the air manifold or tail cap as the jet outlet, which passes the tail air through the source of the pollutant and transfers the pollutant gases to the suction hood. The airflow of the blowing is provided by a blower. The suction section also includes a drain hood which capture and expel the tail air along with contaminants. Numerous studies have been performed on blowing-suction ventilation systems numerically and in the past. In a study, Wang et al. [5] investigated the mixing characteristics of different blowing airflow profiles and changes in the direction of blowing flow in a two-dimensional and laboratory numerical study. Using equation of species transfer and equations of energy, momentum, continuity and mass survival, they simulated the mixing of ambient air with the exhaust air of the jet and obtained the amount of mixing of ambient air with the jet air under the above conditions. In another study, Yang et al. [6] studied the flow ratios (jet-blowing-to-suction ratio) in the design of a blowing-suction ventilation system. They have studied the effect of different flow ratios on the efficiency of a blowing-suction system with the mass fraction of pollutants in a working area. David K. W. [7] has also done his research on the effect of strong jet blowing on the ratio of flow and mixing of blowing air with ambient air. R. Houg et al. [8] also investigated the aerodynamic characteristics in the design of a blowing-suction ventilation system. They have studied its effect on flow lines by increasing the suction hood flow rate. In a study, Marzal [9] studied the effect of jet blowing direction on the efficiency of a blowing-suction system. They used to change the direction of the tail direction relative to the surface of the tank. Wu et al. [10] investigated the impact of air supply direction on the downstream flow field of jet flow. Mi et al. [11] studied the mixing characteristics of the jet flow under a “top hat” velocity profile and a fully developed inner pipe velocity profile. Ferdman et al. [12] investigated the near-field and far-field flow characteristics of circular jets using two non-uniform velocity profiles. Moosae et al [13, 14] proposed an algebraic closure model of microfiber additives in a turbulent channel flow which accounts for the Brownian motion of these fibers. Their study showed that the non-Brownian fibers exhibited maximum

* Corresponding author: khayyamirhossein@gmail.com

\textsuperscript{\#} This is an open access article under the CC-BY license (https://creativecommons.org/licenses/by/4.0/).

© Authors retain all copyrights.
drag reduction near the wall. Moreover, their algebraic closure model showed promising results with a CPU cost lower than the direct Monte-Carlo approach. Zhou et al. [15] used the field synergy theory to study the velocity field and the entrainment of the jet under the three different jet outlet profile conditions. Minkowycz [16] investigated the impact of the turbulence intensity at the jet outlet on laminar flow transition between parallel plates. Chern and Ma [17] also studied a push-pull ventilation system and gave some recommendations about this ventilating system. About 98% of the contaminant generated can be captured by a push-pull system using the proper flow rates [18].

In this research, a blowing-suction ventilation system under the influence of different flow ratios, different directions of blowing flow and their effect on the mixing of ambient air with blowing air is investigated. The above data provides designers with information on the extent of flows and how they are mixed, which is very important for analyzing such systems.

2. Mathematical model

2.1. Problem geometry

The studied geometry is shown in Figure 1. In order to reduce the volume of calculations with the aim of investigating the effect of different inlet jet angles and different flow ratios, the two-dimensional fluid flow assumption has been used, which is an acceptable assumption in this case [2, 7]. According to the designed geometry, the height of the jet and hood is equal to b = 0.2 m and the height of the flange is equal to F = 0.5 m. Also, the distance between the jet and the hood is L = 1.6 m and the solution range is 6m × 3m. The coordinates are located in the middle of the jet outlet.

![Figure 1. Studied geometry schematic](image)

2.2. Governing Equations

Due to the low flow velocities in this problem, the non-compressible flow assumption can be used to solve the comparative mass conservation equations [19], Navier-Stokes, energy, and species transfer [20]. In this modeling, two types of fluids are considered, which are blowing air and drawn air. In fact, the blowing air is the air that is blown by the jet on the source of the pollutant, and the drawn air is the part of the ambient air that is drawn by the jet and transferred to the discharge hood. Blowing air and drawn air are two fluids with different sources but the same physical properties. One of the parameters affecting the performance of this type of ventilation system is the flow ratio (K), which is obtained from the ratio of the output jet flow rate to the inlet air flow rate to the hood (Equation 1).[21]

\[
K = \frac{Q_{in}}{Q_{out}}
\]

Where \( Q_{in} \) is the volumetric flow rate of the output current from the jet (m³/sec) and \( Q_{out} \) is the volumetric flow rate of the input flow to the discharge cap (m³/sec). The mixing rate (\( \varepsilon \)) which is the main indicator for measuring the effect of the flow ratio and the direction of the blow on the blow-suction ventilation system in this study, is defined as the ratio of the mass flow rate of the drawn air to the mass flow rate of the blow air (Equation 3).
\[ \bar{\varepsilon} = \frac{\dot{m}_{ai}}{\dot{m}_{in}} = \frac{\int_{0}^{y_0} \rho \varepsilon \, dy}{\rho Q_{in}} \]  

Equation (2)

In Equation (3), \( \dot{m}_{ai} \) represents the mass flow rate of drawn air \((\frac{kg}{s})\), \( \dot{m}_{in} \) represents the mass flow rate of blown air \((\frac{kg}{s})\), and \( \rho \) represents the density of air \((\frac{kg}{m^3})\). Equation (2) shows the mixing fraction obtained by multiplying the mass fraction of the drawn air, \( Y_a \), by the velocity in the flow path, \( u \) \((\frac{m}{s})\).

2.3. Boundary Conditions

Due to the fact that the inlet velocity in the tail section and the outlet velocity in the suction section are considered to be certain values, the velocity inlet and velocity outlet boundary conditions have been used for these two valves, respectively. The upper boundary and the left boundary in the solution domain use the Pressure inlet condition, and for the right boundary of the solution domain the boundary condition Pressure outlet is used, and the rest, such as the work surface and flange, as a wall with the non-slip boundary condition are considered. The mass fraction of the drawn air, \( Y_a \), is zero at the outlet of the jet and equal to one at the other limits.

2.4. CFD method

To solve the equations governing the problem in the given geometry and according to the boundary conditions, the well-known ANSYS Fluent 18.2 commercial software has been used. According to the speed range studied, the flow is considered turbulent and the turbulent flow simulation is performed using the \( K-\varepsilon \) model [22]. The simple algorithm is used to solve the coupled velocity and pressure [23]. The accuracy of this modeling (residuals) is \( 10^{-5} \).

2.5. Mesh independency

The grid study has been done in order to select the appropriate mesh using three meshing models with different number of cells. To compare the generated cell zones, the linear velocities from the center of the jet to the center of the hood are compared for a flow ratio of \( k = 1 / 2.5 \) (Figure 2). In order to reduce the time of calculations and maintain the accuracy of the calculations, the second network with 88573 cells has been used.

| Generated mesh No. | Number of cells |
|--------------------|-----------------|
| 1                  | 31744           |
| 2                  | 88573           |
| 3                  | 159049          |

Table 1. Number of cells for different generated mesh models

![Figure 2. Central velocity for different cell number](image)
2.6. Validation

Before reviewing the results of the analysis, it is necessary to ensure the accuracy of the simulation performed. One way to validate the results of numerical solution is to check the airflow velocity between the jet and the hood and compare it with the results of Wang et al. [5]. A comparison of airflow velocity at $x/b = 0.5$ obtained in the present study and the results of Wang et al. is shown in Figure 2. The excellent agreement of the results of the present study with the results of Wang et al. indicates the accuracy of the simulations performed in this research.

![Figure 2. Comparison of airflow velocity](image)

3. Results

After verifying and ensuring the performance of the simulations performed, it is time to present and review the results for two different geometries with and without flanges. To study the efficacy of the suction system the 2D stream lines plotted in velocity contours, mass fraction contours also utilized.

3.1. Ventilation with Flange

In this section, the results of the effect of flow ratio and then the effect of blowing angle on the performance of the blowing-suction system are presented. One of the parameters affecting the performance of the blow-suction system is the flow ratio, $K$, which is defined as the ratio of the output jet flow rate to the inlet air flow rate. In the following, the effect of different but widely used flow ratios in the design of such systems is investigated. Figure 4 shows the velocity contours under different flow ratios. By following the stream lines and examining the velocity contour at $K = 1/1$, it is clear that the hood is not able to suck all the tail air, and as a result, part of the tail air is distributed in the environment. By increasing the amount of suction and as a result $K < 1/1$, more amount of blowing air and ambient air is sucked and this prevents the flow of blowing in the working environment.

Dimensionless diagrams of velocity relative to jet velocity in three different sections are shown in Figure 5. Halfway between the tail valve and the suction valve, $x/b < 4$, the hood suction has almost no effect on air velocity, but for $x/b > 4$ the air velocity gradually increases with increasing suction velocity and the ambient air is more under Suction effect. This increases the mixing rate in this area.

Figure 6 shows the mass fraction contours of the tail air. According to Figure 6, it can be seen that for the flow ratio $K = 1/1$, part of the tail air is dispersed in the environment due to insufficient suction, which causes the spread of pollutants in the work environment. As can be seen from Figure 6, a reduction in the flow ratio leads to a significant reduction in the airflow of the tail and thus to pollution in the surrounding environment. The amount of mixing, which actually indicates the ratio of the mass flow rate of the drawn air to the mass flow rate of the blowing air and is presented in Equation (2), is the main measurement criterion in such systems, so that higher mixing rate indicates better system performance.
Figure 4. Velocity contours for different flow ratios
Blowing angle: 0 °, blowing air speed: 2 m/s, flow ratio: A) K = 1/1, B) K = 1 / 1.5, C) K = 1/2, D) K = 1 / 2.5.

Figure 5. Dimensionless velocity diagrams in three different sections a) x / b = 0.5, b) x / b = 4, c) x / b = 7.5.

The amount of mixing in 16 different sections between the blowing and suction valves is calculated and shown in Figure 7. The drawn air is the part of the ambient air that is affected by the jet and suction of the hood and has a non-zero velocity. From the diagram it can be seen that up to halfway, x / b <4, the mixing rate increases linearly for all flow ratios, but at x / b> 4 the situation is different. At a flow ratio of K = 1/1, due to insufficient suction, a very small amount of ambient air is affected by the blowing air, and as it approaches the suction valve, it is observed that the mixing rate tends to zero, which indicates efficiency. The weak system is in the exit of polluted air from the environment in this flow
ratio. With increasing suction and thus decreasing the flow ratio to $K = 1/1.5$, we see a significant increase in the amount of mixing, especially near the suction section. With further reduction of the control flow ratio, the exponential growth of the mixing can be seen by moving towards the suction section.

After examining the effect of the flow ratio, it is time to investigate the effect of different angles of the blowing flow relative to the horizon surface on the mixing rate. The effect of blowing directions of $45^\circ$, $20^\circ$, $0^\circ$, $-20^\circ$, $-45^\circ$ on the mixing rate of the blowing-suction system with a flow ratio of $k = 1/2.5$ is shown in Figure 8. As can be seen from Figure 8, decreasing the blowing angle leads to increasing the mixing rate. Therefore, by adjusting the blowing angle to the lower wall, the performance of the blowing-suction system can be improved.

![Figure 6. Mass fraction of blowing air under different flow ratios.](image)

(a) $K = 1/1$, b) $K = 1/1.5$, c) $K = 1/2$, d) $K = 1/2.5$.

![Figure 7. Mixing rate under different flow ratios](image)
3.2. Ventilation without Flange

In this section the studied ventilation system simulated without the presence of the flange. In Fig. 9 velocity contours and streamlines are shown for different flow ratios. As it can be seen, stream lines in $K = 1/1$ pass the hood and this show that the hood is not effective enough to suction the whole blowing air. As a result, a portion of the air, distribute in the environment. This means that this flow ratio is in appropriate especially when the system is dealing with pollutant gas and materials. By decrease in $K$ or increase in suction power, the performance of the suction system improves and this system doesn’t allow the particles to distribute in the air.

Dimensionless velocity curves are shown in Fig.10. As it given in this figure, the hood suction does not have a considerable effect on the air velocity from the $\frac{x}{b} = 0$ to $\frac{x}{b} < 4$, but for $\frac{x}{b} > 4$ air velocity increases due to the increase in suction velocity and the environment air is affected by suction section more than $\frac{x}{b} < 4$. A noticeable point in Fig 5 c in compare to Fig. 10 c is that when flange deleted from the simulated domain, speed experienced higher speed in compare to the condition that flange exists which can be noticed as a proven effect of flange that prevent solid particles to move up from the earth.
Figure 10. Dimensionless velocity diagrams in three different sections a) $x / b = 0.5$, b) $x / b = 4$, c) $x / b = 7.5$

Mass fraction contour for different flow ratios at $0^\circ$ is given in Fig.11. As it shown in this figure, the system which deal with $K = 1/1$ doesn’t have an acceptable performance because a considerable amount of the air is not affected by suction section which shows the low efficacy of the suction section in this condition. In contrast, when the flow ration decreases, the suction performance improves significantly. This means that by decreasing flow ratio, system is able to ventilate the environment more effective and it is needless to say that most of the discharged air and particles will go to the suction section.

Mixing rate curves for different flow ratios are given in Fig 12. to draw the mentioned curves, 16 different points selected to measure the mixing rate it is obvious that the mixing rate for different flow ratios reached the same value from $x/b = 0$ to $x/b < 6$. While after that the difference in amount of the mixing rate increases significantly which was shown in Fig.10.

Mass fraction contour for different discharge angles is shown in Fig.13. as it can be seen increase or decrease in blowing angle doesn’t play an effective role in the suction section and the suction performance due to its flow ration ($k = 1/2.5$) shows an acceptable performance. It is worthwhile to say that, positive angles cause an undesirable condition near the blowing section because the height if the flow increases by increase the blowing angle.

Figure 11: Mass fraction of blowing air under different flow ratios without flange a) $K = 1/1$, b) $K = 1 / 1.5$, c) $K = 1/2$, d) $K = 1 / 2.5$
Figure 12. Mixing rate under different flow ratios without flange

Figure 13. Mass fraction of blowing air under different blowing angle and constant flow ratio ($K = 1/2.5$)

a) $0^\circ$, b) $20^\circ$, c) $-20^\circ$, d) $45^\circ$ and e) $-45^\circ$
After the numerical investigation on effect of the blowing angle without the presence of the flange, the mixing rate curves for blowing angles $-45^\circ, -20^\circ, 0^\circ, 20^\circ, 45^\circ$ is given in Fig.14. As it can be seen by fixing the blowing angle at the lower angle can help the suction performance.

4. Conclusions

In this study, the effect of flow ratio and blowing angle on contours and velocity diagrams, blowing air mass fraction, and mixing rate, which indicates the mixing of ambient air with blowing air, is investigated for conditions that flange exists and doesn’t exist. The results showed that reducing the flow ratio increases the mixing rate and thus the effectiveness of the blow-suction system and also shows how increasing the suction flow changes the deformation structure of the air flow, reduces the dispersion of the blowing air and prevents Pollutants are released into the environment. Also, the direction of jet air blowing was considered as the second parameter affecting the performance of the system. The results obtained in this section showed that negative angles increase the air mixing and also prevent the diffusion of blown air in the environment and thus the spread of pollution, which is in agreement with the experimental results [4]. Deleting the flange, didn’t cause considerable improvement in the studied system performance. Flangeless geometry only has better performance for $K = 1/1$ flow ratio in comparison to the condition that ventilating system has flange.

References

[1] R.T. Hughes. An Overview of Push—Pull Ventilation Characteristics. Applied Occupational and Environmental Hygiene. 5 (1990) 156-61.

[2] V. Betta, F. Cascetta, A. Palombo. Push-pull ventilation system: A CFD approach for the performance analysis. International Journal of Ambient Energy. 28 (2007) 123-34.

[3] T. Iwasaki, Y. Fujishiro, Y. Kubota, J. Ojima, N. Shibata. Some engineering countermeasures to reduce exposure to welding fumes and gases avoiding occurrence of blow holes in welded material. Industrial health. 43 (2005) 351-7.

[4] A.C.o.G.I.H.C.o.I. Ventilation. Industrial ventilation: a manual of recommended practice. American Conference of Governmental Industrial Hygienists1995.

[5] Y. Wang, M. Quan, Y. Zhou. Effect of velocity non-uniformity of supply air on the mixing characteristics of push-pull ventilation systems. Energy. 187 (2019) 115962.

[6] S. Long, B.S. Dhillon. Proceedings of the 15th International Conference on Man–Machine–Environment System Engineering. Springer2015.

[7] D.K. Cheong, S. Huang. Performance evaluation of personalized ventilation system with two types of air terminal devices coupled with displacement ventilation in a mock-up office. HVAC&R Research. 19 (2013) 974-85.
[8] R. Huang, S. Lin, S.-Y. Jan, R. Hsieh, Y.-K. Chen, C.-W. Chen, et al. Aerodynamic characteristics and design guidelines of push–pull ventilation systems. Annals of occupational hygiene. 49 (2005) 1-15.

[9] F. Marzal, E. Gonzalez, A. Minana, A. Baeza. Influence of push element geometry on the capture efficiency of push–pull ventilation systems in surface treatment tanks. Annals of occupational hygiene. 46 (2002) 383-93.

[10] J. Liu, Y. Wu. Experimental study on airflow fluctuation characteristic of an underfloor air supply terminal unit. Energy and buildings. 42 (2010) 2151-8.

[11] J. Mi, D. Nobes, G. Nathan. Influence of jet exit conditions on the passive scalar field of an axisymmetric free jet. Journal of Fluid Mechanics. 432 (2001) 91-125.

[12] E. Ferdman, M. Otugen, S. Kim. Effect of initial velocity profile on the development of round jets. Journal of Propulsion and Power. 16 (2000) 676-86.

[13] A. Moosaie, N. Shekouhi, A. Ahmadi. Large-eddy simulation of fiber-induced turbulent drag reduction in a channel flow at \( \text{Re}_\tau = 180 \).

[14] A. Moosaie, N. Shekouhi, N. Nouri, M. Manhart. An algebraic closure model for the DNS of turbulent drag reduction by Brownian microfiber additives in a channel flow. Journal of Non-Newtonian Fluid Mechanics. 226 (2015) 60-6.

[15] Y. Zhou, H. Zhu, M. Wang, M. Wang, Y. Wang. Entrainment Analysis Based on the Field Synergy Principle and Air Terminal Device Design. Procedia Engineering. 205 (2017) 1718-24.

[16] W. Minkowycz, J. Abraham, E.M. Sparrow. Numerical simulation of laminar breakdown and subsequent intermittent and turbulent flow in parallel-plate channels: Effects of inlet velocity profile and turbulence intensity. International Journal of Heat and Mass Transfer. 52 (2009) 4040-6.

[17] M.-J. Chern, C.-H. Ma. Numerical investigation and recommendations for push-pull ventilation systems. Journal of occupational and environmental hygiene. 4 (2007) 184-97.

[18] R. Hughes, D. Huebener. Development of Push–Pull Ventilation Criteria. Cincinnati, OH: National Institute for Occupational Safety and Health. (1982).

[19] C.K. Batchelor, G. Batchelor. An introduction to fluid dynamics. Cambridge university press2000.

[20] Y. Zhou, M. Wang, Y. Wang, H. Cui, J. Wang. Development of self-label method to distinguish supply air from room air without tracer in numerical simulations. Building and Environment. 145 (2018) 223-33.

[21] S. Liang, J. Chen, B. Yang, M. Lin, L. Liu, T. Zhang. Research on ventilation antivirus technology in a washing board room based on numerical simulation. International Conference on Man-Machine-Environment System Engineering. Springer2017. pp. 487-94.

[22] Q. Chen. Comparison of different \( k-\varepsilon \) models for indoor air flow computations. Numerical Heat Transfer, Part B Fundamentals. 28 (1995) 353-69.

[23] O. Rouaud, M. Havet. Computation of the airflow in a pilot scale clean room using \( k-\varepsilon \) turbulence models. International Journal of Refrigeration. 25 (2002) 351-61.