Influence of geometrical parameters of air inlet hole on the kinematic characteristics of jet

Juliya Kareeva, Ksenia Bliznyakova, Diana Ashadullina and Raushan Zakieva

1 Kazan State University of Architecture and Engineering, Kazan, Russia
E-mail: jkareeva2503@gmail.com

Abstract. While considering air exchange scheme in rooms with heat dissipations, it is often preferred when jet flows directly into the working area. It is necessary to take into account the influence of irregularities of velocity profiles in jets that are formed at the outflow from nozzle with different geometry. The jet outflow from inlet holes located beyond elbow was numerically studied. The influence of geometrical parameters of inlet hole on the jet characteristics was investigated.

The characteristic flow streamlines for several geometries of a horizontal nozzle, the main characteristics of jet outflow were computed. The relationship between the inlet hole location relative to elbow and the outflow jet characteristics were determined. It was revealed that when the inlet hole is located in close proximity to the elbow, the velocity profiles have strong irregularities, and the jet outflows at an angle to the horizon. When the distance between the hole and the elbow increases, the jet strengthens, the transversal component of velocity does not affect the jet, the outflow angle tends to horizontal.

The obtained results can be used for calculating the circulation of air masses in ventilated rooms.

Key words: numerical methods, flow characteristics, velocity profile, vortex zone, computational fluid dynamics.

1 Introductions

Jet flows in the premises of buildings for various purposes are studied in vast variety of works. Different variants of air distribution schemes in rooms are studied in [1-4]. While considering air exchange scheme in rooms with heat dissipations, it is often preferred when jet flows directly into the working area, for example, spot cooling systems for workplace.

The experimental and theoretical studies determine the relationships between characteristics (jet width and range, longitudinal velocity profiles, axial velocity, average velocity of the return flow, flow velocities in sections of forward and reverse flows, distribution of static pressure along the jet length) of isothermal [5-11] and non-isothermal [12-16] jet flows and the location of inlet and outlet holes.

The characteristics of air masses motion in ventilated rooms are calculated based on the laws of jet flows. In this case, as a rule, the laws of free jets are used to calculate velocity \( u_{ax} \) (Eq. 1) and excess temperature \( \Delta t_{ax} \) (Eq. 2) on the axis:

\[
\frac{u_{ax} = m_{u} \sqrt{h}}{s} \quad (1)
\]

where \( m \) is the kinematic coefficient of the inlet hole (nozzle), \( u_{o} \) is the area-average flow velocity, \( h \) is the slot width, \( s \) is the distance to the cross section in which the axial characteristics of jet are determined:
$$\Delta T = \frac{n \Delta t \sqrt{\Delta T}}{\sqrt{\Delta t}}$$

where $n$ is the heat coefficient of the inlet hole, $\Delta T_o$ is the temperature at the outflow.

Since it is difficult to measure directly the velocity and temperature fields at the nozzle exit, the coefficients $m$ and $n$ are usually determined by measuring the axial characteristics in the main section of jet that leads to inaccuracies of the calculation in some cases. So often we are dealing with jet flows out of a hole of a finite size. Its geometric characteristics have a significant impact on its spreading in the room. Inlet hole in premises of buildings for various purposes can be located in such a way that the working area is located in the zone of initial jet section. Here it is impossible to determine the main characteristics using the known formulas. The only possible way is numerical simulation using computational fluid dynamics (CFD) programs [17-21].

In [22], a method for determining the kinematic and thermal coefficients of the inlet nozzles was proposed. It was illustrated by calculation example for a jet flow from a slot hole located in the duct wall in a series of subsequently located slots. The non-uniformity of velocity fields and static pressure of the excess outflow temperature are taken into account. This article discusses the influence of the distance from the elbow to the inlet hole on the jet characteristics.

2 Material and methods
This article discusses flat jets flowing out of a hole of a finite size $CD (b_0)$, located at various distances from the elbow $ED (\bar{l} = l/b_0)$ (figure 1). The results of numerical calculation are presented below.

![Figure 1. The flow streamlines: a) $\bar{l} = 1.25$; b) $\bar{l} = 5.0$.](image)

The problem is solved numerically using the Fluent software package. The system of differential equations of turbulent motion is closed using the “standard” $k-\varepsilon$ model ($k$ is the kinetic energy of turbulent pulsations, $\varepsilon$ is the specific dissipation of turbulent energy). For modeling the boundary layer near impermeable surfaces, the Standard Wall Function was adopted.

The region geometry is shown in Fig. 1a, b. The following border conditions are adopted:
- AB-Inlet hole, velocity is uniform and directed along the normal to the border: $u_0 = const$; $u_0 = u_0$; $k = 0$, $\varepsilon = 0$;
- BF, FG, GH, HI - free borders, excess pressure $\Delta p = 0$; velocity is directed along the normal to the border $\bar{u} = u_0$, $\frac{dk}{dn} = 0$, $\frac{d\varepsilon}{dn} = 0$;
- BC-AD are impermeable walls $u = 0$, $\frac{du_0}{dn} = 0$;

where $\frac{d}{dn}$ is the normal derivative to the boundary.
In all numerical experiments the following parameters are taken: the duct size is constant and equal to \( b_0 = 0.2 \, \text{m} \), velocity is \( u_0 = 3 \). The distance from the turn to the outflow (ED) varied from 1.25 to 5. Density of supply air and ambient air in environment is accepted \( \rho = 1,225 \, \text{kg/m}^3 \).

3. Results and discussion

Figure 1 shows the flow streamlines. In the first case (figure 1a), inlet hole is located as close as possible to the elbow (\( \bar{l} = 1.25 \)), in the second case \( \bar{l} = 5.0 \). The obtained flow streamlines indicate that at a small distance from the elbow, a vortex zone forms the outlet, which compresses the jet and the outflow occurs at a certain angle. When the distance between the hole and the elbow increases (\( \bar{l} > 3.0 \)), the jet aligns along the duct cross section, the outflow angle tends to 0.

Figure 2 shows the fields of relative components of velocity \( \bar{u} = u/u_0 \) and static pressure \( \bar{P}_s = P_s/P_0 \) at the outflow (\( P_0 = \frac{1}{2} u_0^2 \) is the dynamic pressure determined from the average jet velocity at the outflow and air density \( \rho = 1,225 \, \text{kg/m}^3 \)), dimensionless coordinate is \( \bar{y} = y/b_0 \). These fields are essentially non-uniform, the static pressure is noticeably different from zero, and these circumstances must be taken into account.

![Figure 2. Profiles of longitudinal velocity \( \bar{u}_x \), transversal velocity \( \bar{u}_y \), static pressure \( \bar{P}_s \) for \( \bar{l} = 1.25 \).](image-url)
Figure 3. Profiles of longitudinal velocity component at the outflow.

Figure 3 shows the profiles of longitudinal velocity component at the outflow as a function of dimensionless parameter $\bar{y}$. Regardless of the length of section after the turn, the profile of longitudinal velocity component remains non-uniform. For $\bar{l} < 3.0$ there exist regions of negative velocity values that indicate the presence of vortex zones in this part of the supply nozzle. When the distance between the hole and the elbow increases, the flow strengthens and velocity profile becomes more uniform.

Figure 4 Profiles of transversal velocity component at the outflow.

Figure 4 shows the change in transversal velocity component at the outflow. The proximity of the air inlet to the turn greatly affects the change in transversal velocity component $\bar{u}_y$. For $\bar{l} < 2.25$ the longitudinal and transversal components of velocity are commensurable. So it is important to consider both components when determining the jet characteristics. At a greater distance from the turn, the value $\bar{u}_y$ tends to 0. At $\bar{l} > 3.0$ the transversal component of velocity can be neglected.
Figure 5 shows the static pressure profile at the outflow. It can be seen that for $\tilde{l} < 2.25$ the velocity profile is strongly non-uniform. There is a region of negative values, due to which discharging is created, air leaks from the outside and vortex zones are formed.

4 Conclusion
The main characteristics of jet at the outflow that was formed from nozzle located directly after the elbow were numerically obtained. It was determined that relative positions of outflow and turn had a strong influence on the flow pattern. When the jet came out of inlet hole located in close proximity to the turn, the jet was strongly deformed. The vortex zone arising after the turn compressed the jet, as a result the outflow occurred at a certain angle. The longitudinal and transversal velocity profiles in this case were commensurable, the static pressure was not equal to 0. These factors must be taken into account when determining the characteristics of air jets in a room. When the distance between the hole and the turn increased, the jet strengthened, the outflow angle tended to 0. The transversal velocity component and static pressure were almost equal to zero and can be neglected.

References
[1] Cehlin M 2006 Visualization of Airflow, Temperature and Concentration Indoors Doctoral thesis University of Gävle, Published in Stockholm.
[2] Sattari A, Fallenius Bengt E G, Fransson Jens H M, Sandberg M 2011 PIV visualization study in a two-dimensional room model with rapid time varying ventilation flow velocities. Proceedings RoomVent 12th International conference on air distribution in rooms, Trondheim, Norway. doi: 10.1080/14733315.2013.11684015
[3] Guangyu C, Kurnitski J, Ruponen M, Seppänen O 2009 Experimental investigation and modeling of a buoyant attached plane jet in a room Applied Thermal Engineering, Elsevier, 29 pp 14-15. doi: 10.1016/j.applthermaleng.2009.01.016
[4] Fillingham P, Novosselov I 2018 Similarity of the Wall Jet Resulting from Planar Underexpanded Impinging Jets. AXiv preprint arXiv: 1812.11220.
[5] Posokhin V, Kareeva J 2016 About pressure losses during jet air supply to limited volumes Izvestiya KGASU, 2 (36), pp 146-151.
[6] Abramovich G 1963 The theory of turbulent jets. MIT Press, Cambridge, MA doi:
10.7551/mitpress/6781.001.0001.

[7] Zou Y 2000 Velocity Decay in Air Jets for HVAC Applications. ASHRAE Transactions, Vol. 106, Part 2.

[8] Zasimova M, Ivanov N 2018 Numerical Simulation of Air Distribution in a Room with a Sidewall Jet under Benchmark Test Conditions. The eighth polyakhov’s reading: Proceedings of the International Scientific Conference on Mechanics 1959(1):050033. doi: 10.1063/1.5034661

[9] Malmstrom T-G, Zou Y 2000 Influence of outlet characteristics on free axial air jets Roomvent, Reading, UK.

[10] Barenblatt G, Chorin A, Prostikishin V 2005 The turbulent wall jet: A triple-layered structure and incomplete similarity Journal of Applied Mathematics, Vol. 102, pp 8850-8853.

[11] Kee-Chiang Chunga, Shou-Ping Hsu 2001 Effect of ventilation pattern on room air and contaminant distribution Building and Environment, Vol. 36, pp 989-998. doi: 10.1016/S0360-1323(00)00051-2

[12] Cao G, Kurniatski J, Ruponen M, Mustakallio P, Seppänen O 2009 Plane-Air-Jet Corner Zone Modelling in a Room Ventilated by an Active Chilled Beam International Journal of Ventilation, Vol. 7, pp 287-298. doi: 10.1080/147333315.2009.11683819

[13] Cao G, Kurniatski J, Ruponen M, Mustakallio P, Seppänen O 2008 Active Chilled Beam Wall Jet Prediction by the Free Convection Model International Journal of Ventilation, Vol. 7, pp 169-178. doi: 10.1080/147333315.2008.11683809

[14] Bing-Chen Wang F, Koupriyanov M, Tully B 2013 Experimental study of a non-isothermal wall jet issued by a displacement ventilation system Building and Environment Vol. 66, pp 131-140. doi: 10.1016/j.buildenv.2013.04.019

[15] Nielsen P 2000 Velocity distribution in a room ventilated by displacement ventilation and wall mounted air terminal devices Energy and Buildings Vol. 31, Iss. 3, pp 179-187. doi: 10.1016/S0378-7788(99)00012-2

[16] Tomasi R, Krajčík M, Simone A, Olesen B 2013 Experimental evaluation of air distribution in mechanically ventilated residential rooms: Thermal comfort and ventilation effectiveness Energy and Buildings, 60, pp 28-37. doi: 10.1016/j.enbuild.2013.01.003

[17] Kazuhide I, Kiao I, Takashi K, Toshikatsu U, Tomoyuki E, Toshiaki O, Hiroki O, Shinsuke K, Koji S, Yoshihide S, Hiroshi M, Hajime Y, Weirong Zh, Jiyuan T 2015 CFD Benchmark Tests for Indoor Environmental Problems: Part 1. Isothermal/Non-Isothermal Flow in 2D and 3D Room Model International Journal of Architectural Engineering Technology 2, pp 1-22. doi: 10.15377/2409-9821.2015.02.01.1

[18] Yongson O, Badruddin I, Zainal Z, Aswatha Narayana P 2007 Airflow analysis in an air conditioning room Building and Environment, 42 (3), pp 1531-1537. doi: 10.1016/j.buildenv.2006.01.002

[19] Nielsen P 2007 Analysis and design of room air distribution systems. HVAC and R Research, 13(6), pp 987-997. doi: 10.1080/10789669.2007.10391466

[20] Nielsen P 2015 Fifty years of CFD for room air distribution Building and Environment Vol. 91, pp 78-90. doi: 10.1016/j.buildenv.2015.02.035

[21] Hyeunguk A, Donghyun R, James L 2017 Ventilation and energy performance of partitioned indoor spaces under mixing and displacement ventilation Building Simulation 11. doi: 10.1007/s12273-017-0410-z

[22] Posokhin V, Varsegova E, Ziganshin A 2017 On the numerical determination of characteristics of the supply nozzle News of the KSUAE, 1 (39) pp 173-179.