CFD-modeling of flow in confluence nodes of ventilation units of multi-storey buildings

Aiman Shopayeva and Rinat Safiullin

1Kazan State University of Architecture and Engineering, Kazan, 420043, Russia
E-mail: safiullin_rinat@mail.ru

Abstract. This paper considers the flow characteristics in ventilation units, which are used in natural ventilation systems of multi-storey residential buildings. Three configurations of ventilation units were considered: standard with straight walls, with a beveled wall and with a guide insert. For each configuration, three sizes of bypass hole at the confluence of flows from the satellite channel and the collection channel were considered. The problem was solved numerically using the CFD Flow3d software package.

It was established that for a ventilation unit with a straight wall in the satellite channel, the formation of separation zones that are significant in width and length at the confluence node is typical. The length and width of separation zones affecting the resistance of ventilation unit as a whole increases with the enlarging of the bypass hole size in the collection channel and decreases when using a beveled end wall, that directs the flow upstream of the exhaust air. For the considered geometries of confluence nodes of ventilation units, the relationships between local resistance coefficients and velocity (Reynolds number) in the collection channel are obtained.

Keywords: natural ventilation, ventilation unit, air flow, numerical simulation, vortex zones, coefficient of local resistance

1 Introduction

The experimental data on the local resistance coefficients $\xi$ (LRC) refer to the fluid movement with a normal aligned velocity field [1]. In practice, local resistances are sometimes placed so close one to another that the flow between them does not have time to equalize, as the vortex formation occurring when passing through the local resistance results in a significant extension downstream. The distance after local resistance, within which a normal (aligned) velocity diagram is established and the local resistance influence on the flow stops, is called the influence zone of local resistance. The total coefficient of local resistance of several closely located resistances can be either greater or less than the sum of the corresponding coefficients of unit resistances depending on the length of the straight section between them [2, 3].

In cases when distance between individual resistances is less than influence zone, the perturbing effect of one local resistance affects another. So, the resistance coefficient of two closely located diaphragms is only 40% of the sum of their resistances [3, 4]. The authors explain the decrease in total coefficient by the fact that the flow after the first resistance does not have time to equalize, and the pressure loss that would be spent on this equalization decreases for large influence zone. In [5], similar cases of mutual influence of local resistances were called “interference”. The interference phenomenon is poorly studied, especially for many "non-standard" nodes of ventilation systems, in which elements are located one after another.
It is important to consider the mutual influence of local resistances when calculating the natural ventilation systems of multi-storey residential buildings with “warm attic”, constructed from ventilation units [6] with one-sided or two-sided arrangement of satellite channels (figure 1). The node, connecting the satellite channel with the collection trunk-channel, has an elbow fitting and a T-junction directly one after another (figure 1 b). Obviously, a simple addition of LRC of these node elements will give a large error. The inaccuracy of LRC calculation can significantly distort the calculation results for the so-called “hybrid” ventilation systems when mechanical traction inducers - ejectors [7, 8] or supply and exhaust units with recuperators [9] are introduced into the systems from ventilation units. Therefore, works in this direction have relevance.

LRC of “non-standard” nodes of ventilation units can be found by modeling using the Computational Fluid Dynamics (CFD) software. The CFD usage allows one to determine the total losses and the total LRC for ventilation network nodes containing several disturbing elements without experiments with bulky concrete blocks produced by the construction industry.

Figure 1. A typical solution for a natural ventilation system made of ventilation units in a multi-storey residential building with a warm attic:

   a) arrangement of floor-by-floor ventilation units and system elements in the attic;

   b) scheme of the ventilation system.

1 – Head (diffuser) of the combined ventilation duct; 2 – Shaft; 3 – Pallet;
4 – Baffle apron made of galvanized steel; 5 – Ventilation unit; 6 – Machine room of elevator.

It would be very useful to apply CFD modeling for calculating the total LRC for the collection channel as a whole, starting from the exhaust grill of the lower floor and ending by the exit from the exhaust shaft. It is also important to determine pressures at the points of branches connection (grills, confluence flows) at all intermediate floors and ways to reduce aerodynamic resistance. The presence of such information will greatly simplify the calculation, which would be reduced to select the necessary characteristics of ventilation units of multi-storey buildings.

Analysis of literature on natural ventilation of multi-storey residential buildings shows that there were no studies of local resistance interference in ventilation units. This work is an attempt to study this...
phenomenon and to search for a method of reducing the resistance in the confluence node. Meanwhile, studies with CFD modeling of flows in individual elements, such as a T-junction, elbow fitting, and T-shaped confluence node, are rather widely presented in works devoted to problems of reducing the resistance of connection nodes to brick channels of kitchen suction [10-16]. In [17-19], the sizes of vortex zones and pressure losses in the straight bend were numerically determined, as well as the length of the influence zone behind this disturbing element, which is very typical for mechanical exhaust ventilation systems. An interesting way of reducing resistance with the use of profiling inserts repeating the contours of vortex zones behind elbow fittings is given [17]. It was noted in [20] that the use of profiling inserts in T-junctions can reduce hydraulic losses by 30%. In [21], the flow is modeled in a “non-standard” U-shaped branch, where formation of large vortex zones one after another is observed.

Analysis of ways to reduce the resistance in confluence nodes of ventilation units allows proposing for consideration options with a beveled wall in the knee of the satellite channel and with a guide insert in the collection channel. These two options are further discussed in the work.

2 Materials and methods
In this paper, three configurations of one-sided ventilation units were considered (figure 2). For each of the selected configurations, three variants of bypass hole sizes \( b \) were calculated at the confluence of flows from the satellite channel and the collection channel. The simulation was carried out for various air velocities at the outlet of ventilation unit (from 0.4 m/s to 2 m/s). The problem is solved numerically using the Flow 3d software package. The system of equations of plane turbulent motion was closed using the standard \( k-\varepsilon \) model. Standard wall functions were used for modeling flow in boundary layers near solid walls.

The following conditions were accepted at the boundaries of the computational domain (figure 3): at the \( AD \) boundary, the velocity in the positive direction of the \( x \) axis is constant and equal to \( v_i \); at the \( HK \) boundary, the velocity in the positive direction of the \( z \) axis is constant and equal to \( v \); \( BC \) is the permeable boundary, the velocity \( v_0 \) at which is determined from the law of conservation of mass; boundaries \( ABL, DEFH, CK \) are solid impermeable walls for which the condition of adhesion (non-slip) is fulfilled. The total number of grid cells in the computational domain is 20 thousand. The number of cells was taken not less than 20 across the width of the satellite channel \( b \), as recommended in [22].

![Configuration of a typical ventilation unit and its design schemes for flow modeling](image)

**Figure 2.** Configuration of a typical ventilation unit and its design schemes for flow modeling: a) standard; b) with a beveled wall; c) with a guide insert.
3 Results and discussion

Numerical simulation was used to determine the velocity, pressure fields, and flow lines that are typical for each configuration of ventilation units shown in figure 2. The results of some numerical experiments performed for different configurations of the computational domain with a change in air velocity at the outlet from the trunk-channel are shown in figures 4, 5 and 7. At the same time, LRC for nodes of confluence flows were calculated. Plots of their changes in a wide range of Reynolds numbers \( \text{Re} = \frac{\nu \cdot b}{\nu} \), \( \nu \) is the kinematic viscosity of air) are shown in figures 6, 8 and 10.

![Figure 3. The computational domain for the flow.](image)

![Figure 4. Velocity fields and flow lines for a “standard” ventilation unit with a bypass hole of \( b=0.2 \) m for \( HK \)-boundary velocities \( v \) equal to: a) 0.4 m/s; b) 2.0 m/s.](image)
Figure 4 shows that for a “standard” ventilation unit with a straight (at an angle of 90°) wall in the satellite channel, the formation of separation zones are significant in width and length at the confluence node is typical. Their dimensions increase with the enlarging of air velocity in the satellite channel. The length and width of the separation zones affecting the resistance of ventilation unit increases with increasing size of the hole \( b \) at the inlet to the trunk-channel. It is seen when comparing the calculation results for ventilation units with bypass hole heights of \( b = 200 \) mm (figure 4) and \( b = 400 \) mm (figure 5). The relationship between LRC and \( Re \) number for various hole widths in the satellite channel (\( b = 200 \) mm; 300 mm, 400 mm) for the “standard” ventilation units is shown in figure 6.

The flow chart at the confluence node of ventilation unit somewhat changes when using a beveled end wall in the satellite channel (figure 2, scheme \( b \)). Figure 7 shows that the width of the separation zone is small, but its length practically does not decrease for air velocities in the satellite channel, which were used to calculate the “standard” ventilation unit. The relationship between LRC and \( Re \) number for various hole widths in the satellite channel (\( b = 200 \) mm; 300 mm, 400 mm) for ventilation units with a beveled wall is shown in figure 8.

The velocity fields and flow lines for ventilation units with a guide insert are shown in figure 9. For them a significant decrease of sizes of separation zones, and, consequently, of their resistances is observed.

The relationship between \( \xi \) and \( Re \) number for the 40° angle of inclination of the guide insert is shown in figure 10. It can be seen that such structural solutions of confluence nodes at high velocities in the trunk-channel are aerodynamically more “perfect” and lead to a decrease of LRC of the entire fan unit.
**Figure 7.** Flow lines typical for a ventilation unit with a beveled wall ($b = 0.3$ m) for $HK$-boundary velocities $v$ equal to: a) 0.4 m/s; b) 1.2 m/s.

**Figure 8.** Relationship between $\xi$ and the Reynolds number for the “standard” ventilation unit with various hole widths in the satellite channel: a) 200 mm; b) 400 mm.

**Figure 9.** Velocity fields and flow lines typical for a ventilation unit with a guide insert ($d = 0.4$ m) for $HK$-boundary velocities $v$ equal to: a) 0.4 m/s; b) 1.2 m/s; c) 2.0 m/s.
As a result of the work, it was established:
- for a “standard” ventilation unit with a straight (at an angle of 90°) wall in the satellite channel, the formation of separation zones that are significant in width and length at the confluence node is typical. Their dimensions increase with increasing air velocity in the satellite channel.
- the length and width of the separation zones affecting the resistance of the ventilation unit as a whole increases with increasing the size of the bypass hole in the collection channel and decreases when using a beveled end wall, that directs the flow upstream of the exhaust air.

For the considered geometries of the confluence nodes of ventilation units, the relationships between local resistance coefficients and velocity (Reynolds number) in collection channel are obtained. The results of numerical experiments allow one to correct the existing methods for calculation ventilation units.

**4 Conclusions**

The LRC values calculated during the study of flows in ventilation units allow one to adjust the existing methods for their calculation. The given data can help the designer in aerodynamic calculation of natural ventilation systems of multi-storey residential buildings, consisting of ventilation units. If one knows the configuration of the selected ventilation unit, the channels geometry and velocity of air inlet into the satellite channel or the trunk-channel on the floor, he will be able to more accurately determine the local resistance coefficients for these elements.

As a result of the work, it was established:
- for a “standard” ventilation unit with a straight (at an angle of 90°) wall in the satellite channel, the formation of separation zones that are significant in width and length at the confluence node is typical. Their dimensions increase with increasing air velocity in the satellite channel.
- the length and width of the separation zones affecting the resistance of the ventilation unit as a whole increases with increasing the size of the bypass hole in the collection channel and decreases when using a beveled end wall, that directs the flow upstream of the exhaust air.

For the considered geometries of the confluence nodes of ventilation units, the relationships between local resistance coefficients and velocity (Reynolds number) in collection channel are obtained. The results of numerical experiments allow one to correct the existing methods for calculation ventilation units.

**References**

[1] Idelchik I E 2017 Flow Resistance: A Design Guide for Engineers. CRC Press.
[2] Vasenin V I, Bogomyagkov A V, Sharov K V 2012 Investigation of local resistances of gating system *Bulletin PNRPU Mechanical engineering, materials science*, pp 46-53.
[3] Levintan E Yu, Sheypak A A 2009 Numerical modeling of the mutual influence of elements of hydraulic systems *Bulletin of Moscow State Industrial University*, vol. 4 (17), pp 50-55.
[4] Vasenin V I, Sharov K V 2015 Determination of the liquid flow rates and pressures of the fluid in the storey shaped gating system. In *5th the International Conference on Science and Technology*, pp 48-70.
[5] Vasenin V I, Bogomyagkov A V, Sharov K V 2013 Investigation of interference of local resistances of the gate system *Bulletin PNRPU. Mechanical engineering, materials science* pp 82-87.
[6] Pukhkal V, Vatin N, Murgul V 2014 Centralized natural exhaust ventilation systems use in
multi-story residential buildings *Applied Mechanics and Materials*, vol. 680 pp 529-533. doi: 10.4028/www.scientific.net/AMM.680.529

[7] Concannon P 2002 Residential Ventilation *AIVC Technical Note* 57, Air Infiltration and Ventilation Center.

[8] Noel O, Le Dean P, Krikorian O, Haller R Natural Ventilation Activated by Induction AIVC 21st Annual Conference Innovations in Technology, The Hague. Netherlands, 26-29 September 2000.

[9] Pukhkal V, Vatin N, Murgul V 2014 Central ventilation system with heat recovery as one of the measures to upgrade energy efficiency of historic buildings *Applied Mechanics and Materials*, vol. 633-634 pp 1077-1081. doi: 10.4028/www.scientific.net/AMM.633-634.1077

[10] Li A, Zhang W, Gao M 2017 Field test and CFD modeling for flow characteristics in central cooking exhaust shaft of a high-rise residential building *Energy and Buildings*, vol. 147, pp. 210-223. doi: 10.1016/j.enbuild.2017.04.056

[11] Zheng Q, Wang H, Chen Q, Zhang O 2014 Numerical simulation on central exhaust system design for residential kitchen in high-rise buildings. Proceedings of the 8th International Symposium on Heating, Ventilation and Air Conditioning *Lecture Notes in Electrical Engineering*, vol. 263, pp 383-390. Springer, Berlin, Heidelberg. doi: 10.1007/978-3-642-39578-9_41

[12] Georgiou M, Papalexandris M V 2017 Turbulent confluence in T-junctions: The role of the temperature as an active scalar *International Journal of Heat and Mass Transfer*, vol. 115, Part B, pp 793-809. doi: 10.1016/j.ijheatmasstransfer.2017.08.081

[13] Li A, Kosonen R 2019 Stack effects of kitchen exhaust shaft in high-rise buildings *Kitchen Pollutants Control and Ventilation*. Springer, Singapore. doi: 10.1007/978-981-13-6496-9_8

[14] Selvam P K, Kulenovic R, Laurien E 2016 Experimental and numerical analyses on the effect of increasing inflow temperatures on the flow confluence behavior in a T-junction *International Journal of Heat and Fluid Flow*, vol. 61, Part B, October 2016, pp 323-342. doi: 10.1016/j.ijheatfluidflow.2016.05.005

[15] Li G, Wang Y, Feng G, Jiang M, Liu Q 2012 Simulation and analysis on operating characteristics of central exhaust system in high-rise residential *Advanced Materials Research*, vol. 614-615, pp 656-664. Trans Tech Publications, Ltd. doi: 10.4028/www.scientific.net/AMR.614-615.656

[16] Yang Q Y, Liu T H, Lu W Z, Wang X K 2013 Numerical simulation of confluence flow in open channel with dynamic meshes techniques *Advances in Mechanical Engineering*, vol. 2013. doi: 10.1155/2013/860431

[17] Ziganshin A M, Beljaeva E E, Logachev K I, Averkova O A 2019 Decrease of pressure losses in elbow fittings of ventilation systems of thermal power plant buildings *IOP Conf. Series: Earth and Environmental Science* 288. doi: 10.1088/1755-1315/288/1/012133

[18] Crawford N M, Cunningham G, Spence S W T 2007 An experimental investigation into the pressure drop for turbulent flow in 90° elbow bends. Proceedings of the Institution of Mechanical Engineers, Part E: *Journal of Process Mechanical Engineering*. doi: 10.1243/0954408JPME84

[19] Zardin B, Cillo G, Borghi M, D'Adamo A, Fontanesi S 2017 Pressure losses in multiple-elbow paths and in v-bends of hydraulic manifolds *Energies*. doi: 10.3390/en10060788

[20] Ziganshin A M, Badikova L N 2017 Numerical simulation of flow in a profiled ventilation confluence T-junction *News of higher educational institutions. Construction*, vol. 6 (702) pp 41-48.

[21] Ziganshin A M, Ozerov A O, Solodova E E 2019 Numerical study of flow in a U-shaped branch and decrease in its resistance *News of higher educational institutions. Construction*, vol. 1 pp 82-85.

[22] 2008 FLOW-3D® User’s Manual, Version 9.3. Flow Science, Inc., 821 p.