Numerical and experimental study of the device for the control of the ventilating airflow

Fanis Safiullin\textsuperscript{1}[0000-0001-5206-5154], Vladimir Broyda\textsuperscript{1}[0000-0002-1472-9295]

\textsuperscript{1}Kazan State University of Architecture and Engineering, Kazan, 420043, Russia
E-mail: fanfagsaf@rambler.ru

Abstract. Ventilation systems often operate under changing conditions, which leads to a deviation of the actual air flow from the calculated one. When the gravitational pressure increases, they remove more air than the calculated amount, which leads to increased heat expenditure on heating the air, which compensates for exhaust during the cold period of the year. To maintain a constant calculated flow rate and reduce heat losses during the cold season, especially for natural exhaust systems, it is rational to use direct-acting ventilation air flow regulators.

The aerodynamic characteristics of such a device for regulating the flow of ventilation air are obtained numerically and experimentally: the coefficient of local resistance $\zeta$ and the parameter $k$, which characterizes the aerodynamic force acting on the movable element of the regulator.

The accuracy of experimental study is proved. Comparison of the results of numerical and experimental research showed their good correspondence, which confirms the suitability of the data obtained for the design and calculation of air flow regulators of the considered type.

Keywords: channel, diaphragm, cylindrical body, flow regulator, pressure, force.

1 Introduction

Ventilation systems are designed to provide the calculated air flow. The calculated flow rate is usually determined by diluting the harmful emissions entering the room in the form of heat, moisture, gases, and dust. In cases when these revenues are not expressed clearly, the design flow is determined by the standards for the supply or removal of air per unit volume of the space (the exchange) or regulation of supply of fresh outdoor air per person, or per unit any equipment (e.g., plumbing).

The operation of ventilation systems is affected by changing conditions: a decrease in outdoor air temperature, especially noticeable in the cold season, increases the gravitational pressure, the value of which significantly affects the performance of natural ventilation systems, which in this case increases if it is not regulated. Wind pressure and changes in its direction also have a noticeable effect on air consumption in such systems. Regulating the air flow or disabling individual sections of the system also changes the air flow in adjacent sections and the air flow of the ventilation system as a whole. Reliable and efficient operation of the ventilation system under changing conditions largely depends on its ability to maintain the calculated air flow through the use of the provided regulating devices.

A lot of research has been devoted to various methods and aspects of regulating ventilation systems, for example [1, 2]. In these works, the regulation is considered due to special inserts and flaps located inside the channel.

The main element for the regulation of the ventilation system is a regulating device. Usually, such a device includes an element that is moved in the channel, in the form of plates that rotate on the axes (various throttle valves, dampers), in the form of plates or a plate that moves perpendicular or at an angle to the channel axis (gates, etc.).

In this paper, we numerically and experimentally investigate the aerodynamics of a cylindrical body moving along the axis of a circular channel with a diaphragm, which is a model of a control device such as a valve with a spool. Such devices are widely used in regulating the flow of liquid and gas in many areas of technology. The movement of a cylindrical body changes the flow pattern, and the velocity and pressure fields near the control device change. A characteristic of the pressure drop in such a control device can be its variable coefficient of local resistance $\zeta$, which depends on the position of the cylindrical body relative to the diaphragm.
In many cases, the value of the force of the aerodynamic impact of the flow on the displaced cylindrical body is important. More often, this force needs to be compensated, but sometimes it is used to move the body that regulates the flow. Such use of this force is assumed in automatic direct-acting regulators [3], such devices can be used to maintain approximately constant air flow under changing conditions, for example, when the gravitational pressure increases in natural exhaust ventilation systems.

A generalized characteristic of the aerodynamic impact force can be the dimensionless parameter $k$, which is equal to the ratio of the pressure drop between the front and rear surfaces of the cylindrical body to the dynamic pressure of the flow.

Modern numerical studies aimed at studying the aerodynamic resistances of various parts of ventilation systems and determining the coefficients of their local resistances $\zeta$ are often based on CFD methods. For example, in [4, 5, 6], a numerical study was conducted to determine the coefficient of local resistance of tees in mechanical ventilation systems. Using CFD modeling, characteristic flow patterns and the dependence of the local resistance coefficient on the branch diameter were obtained. In works [7, 8, 9], CFD modeling allows you to simulate the air flow in a channel or room in different speed ranges.

As a rule, such modeling is based on modern semi-empirical models of turbulence, often using the $k$-$\varepsilon$ model [10, 11, 12]. In these works, the implemented $k$-$\varepsilon$ model provides improved predictions for determining the flow rate. In such numerical studies, it is possible to calculate the force effect of a liquid flow [13] or gas [14] on the device elements and calculate the impact parameter $k$.

Traditionally, the actual characteristics of regulating devices are found, refined and verified in experimental studies [15,16,17], which are carried out on aerodynamic stands, in wind tunnels, hydrolothes and other similar devices.

A number of methods and devices for regulating, limiting and stabilizing air flow in ventilation systems are known [18, 19], in which balancing is performed using dampers and barriers. For example, in [20], thanks to the dynamic regulation of the dampers and the fan speed, the necessary amount of air is distributed efficiently. In addition, ventilation systems can be regulated by resetting the static pressure settings [21], or by changing the concentration of carbon dioxide, depending on time [22].

As a rule, such control devices have a significant initial aerodynamic resistance, are mainly intended for use in mechanical ventilation systems and are used only in limited ways.

2 Materials and methods
A regulating device consisting of a cylindrical body moving along the axis of a circular channel relative to the diaphragm installed in the cross-section of the channel is considered.

The aim of the study is to determine the numerical and experimental dependences of local resistance coefficients $\zeta$ and aerodynamic parameters $k$ that characterize the effect of air flow on a cylindrical body when it moves along the axis of the circular channel relative to the diaphragm for several variants of the length of the body being moved. Such data is required when developing the design and design of regulating devices of this type.

Based on numerical simulation, the velocity and pressure fields are calculated, which are used to calculate the corresponding aerodynamic characteristics $\zeta$ and $k$.

The features of numerical modeling, such as the turbulence model (in this work, the standard $k$-$\varepsilon$ turbulence model was used), the method for setting boundary conditions, air properties, grid adaptation techniques, etc. were verified, as in [23], by comparing the results of numerical calculation with a reliable, for example, experimentally obtained value of the local resistance coefficient of the previously studied device. The numerical solution should not depend on the degree of grinding of the calculated grid. The control parameter that determines the convergence of the simulation result for such problems can be, for example, the pressure drop on the calculated section of the channel with the device under study.
In this study, the object for verification of the numerical calculation method is a channel with a diameter of 0.16 m, a length of 3.35 m with a flat diaphragm having a hole in the center with a diameter of 0.11 m. In such a channel, the air flow was numerically modelled at an average speed of 7 m/s at the inlet. The pressure drop in the calculated section of the channel with the diaphragm was used as a control parameter. As a result, the total pressure at the entrance to the channel $P_1=170.59$ Pa, at the exit from the channel with a diaphragm $P_2=31.15$ Pa, and the average dynamic pressure in the channel $P_d=30.01$ Pa were determined numerically under the specified conditions. The pressure drop in the same channel without a diaphragm is $\Delta P=17.82$ Pa. The local resistance coefficient of the diaphragm $\zeta$ is calculated using the formula:

$$\zeta = (P_1 - P_2 - \Delta P)/P_d$$

$$\zeta = (170.59 - 31.15 - 17.82)/30.01 = 4.05$$

The reference (experimental) value of the local resistance coefficient of a flat diaphragm corresponding to the same conditions $\zeta = 3.75$, the difference is about 8%, which can be considered a good confirmation of the settings of the chosen numerical model.

Diagram of the problem of flow around a cylindrical body in a circular channel with a diaphragm (Figure 1):

![Diagram of the problem](image)

Figure 1. Scheme of the problem to be solved: 1 - the wall of a circular channel $d=0.16$ m, 2 - a movable cylindrical body with a diameter $d_b=0.11$ m, 3 - a flat diaphragm with a hole diameter $d_d=0.11$ m, l - a variable distance from the cylindrical body to the diaphragm.

The calculations considered two variants of the length of the cylindrical body $l_b=0.10$ m and $l_b=0.05$ m. The numerical study was performed at variable values of the distance $l$ in the range from 0.0048 to 0.05 m.

The characteristic calculated flow pattern in the form of current lines (Figure 2):

![Figure 2](image)

Figure 2. The calculated flow pattern under the conditions: the average air velocity in the channel is 7 m/s, the length of the cylindrical body $l_b=0.10$ m, the distance $l=0.05$ m, the number $Re=66000$.

Areas of increased flow velocity in the areas of narrowing, vortex zones behind the cylindrical body and in the wall area after the diaphragm are clearly visible.

As a result of calculations, total pressures were determined before the cylindrical body and after the diaphragm in sections 1-1 and 2-2 (Figure 3a), which are located at a sufficient distance from the cylindrical body $(15d)$ and at a distance $(20d)$ after the diaphragm, as well as the pressure on the front and rear walls of the cylinder at a variable distance $l$ (Figure 3b).
Based on the calculated average pressures in the cross sections, the values of the local resistance coefficient $\zeta$ were determined using the formula:

$$\zeta = \frac{(P_{1,1} - P_{2,2}) - \Delta P}{P_{d}}$$

(2)

where $P_{1,1}$ is the mean value of the total pressure in the cross-section 1-1, Pa; $P_{2,2}$ is the average value of total pressure in the cross-section 2-2, Pa; $\Delta P$ is the drop in full pressure in the channel distance between cross sections 1-1 and 2-2 in the absence of the cylindrical body and the diaphragm, Pa; $P_{d}$ is the average dynamic pressure in the channel cross section, Pa.

![Figure 3. Pressure diagrams for the cylindrical body length variant](image)

Figure 3. Pressure diagrams for the cylindrical body length variant $l_b=0.10$ m and $l=0.05$ m from the diaphragm: (a) 1-in section 1-1, 2-in section 2-2, (b) 3-on the front wall of the cylindrical body, 4-on the back wall of the cylindrical body, $r$-transverse coordinate.

The parameter $k$, which characterizes the aerodynamic force of impact on a cylindrical body, is calculated using the formula:

$$k = \frac{(P_f - P_b)}{P_d}$$

(3)

where $P_f$, $P_b$ - the average pressure values, respectively, on the front and back surfaces of the cylindrical body, Pa.

As a result, the values $\zeta$ (Figure 5), and $k$ (Figure 6) depending on the relative size of the $l/d$ for the calculated variants of the problem.

For the considered variants, the general trends of changes in the characteristics of $\zeta$ and $k$ are preserved, and the values of $\zeta$ and $k$ increase with a decrease in $l/d$. The values $\zeta$ and $k$ for a long and shortened cylindrical body are quite close, but the device with a shortened cylinder will be slightly smaller.

Obviously, the results obtained apply to all such devices with geometrically similar dimensions in the turbulent mode of air movement. The dependencies $\zeta$ and $k$ on $l/d$ can be approximated by power functions:

$$\zeta = a_1 \cdot (l/d)^{b_1}$$

$$k = a_2 \cdot (l/d)^{b_2}$$

(4)

During the experimental study, the aerodynamic characteristics of the flow in the channel were determined on the stand, which were used to calculate the values $\zeta$ and $k$ of the control device corresponding to different positions of the mobile control body relative to the diaphragm. Then these characteristics were compared with the results of numerical modelling.

An experimental research bench was designed and constructed (Figure 4), which can be used to study the properties of the regulating device. In addition, on the same stand, you can check the
performance of the regulator to maintain a constant flow rate when modelling changes in gravitational pressure. To solve this problem, there are: an additional throttle valve 13, an additional micromanometer 9, with which you can simulate and control the change in gravitational pressure that occurs, when the temperature of the outside air changes.

Figure 4. Scheme of the experimental stand: 1 – air duct \( d=160 \text{ mm} \); 2 – model of the regulating device; 3 – two-channel pneumatic tube; 4 – throttle-valve \( d=160 \text{ mm} \); 5 – static pressure chamber; 6 – throttle-valve \( d=315 \text{ mm} \); 7 – fan; 8 – air duct \( d=315 \text{ mm} \); 9 – fitting; 10 – differential pressure gauge TESTO 510; 11 – dynamometer; 12 – viewing window; 13 – throttle-valve \( d=160 \text{ mm} \); 14 – thread; 15 – block; 16 – movable element 100 mm long; 17 – diaphragm 110 mm in diameter; 18 – movement limiter; 19 – guides; 20 – cross-section for air passage.

The measurement of dynamic pressures in the air duct 1, registered by a differential pressure gauge 10, was carried out on an aligned section of sufficient length (15 calibers) using a two-channel pneumatic tube 3, installed along the axis of the air duct 1. The air velocity and flow rate in the experiments were calculated from the found dynamic pressures. Depending on the position of the movable element 16 of the regulating device 2, which was determined visually by a linear scale installed on the guide 19 through a glazed viewing window 12 in the wall of the air duct, the difference in static pressures taken from the fittings 9 at the boundaries of the measuring section length \( l_m \) was determined. Based on the measured static pressure drops, taking into account the linear pressure loss \( R \) in the area between the fittings, determined by reference tables, the local resistance coefficients \( \zeta \) corresponding to the position of the mobile element were calculated.

The forces from the influence of the air flow on the movable element of the regulating device were measured using a spring dynamometer 11, connected to the movable element by a thread 14, thrown over the block 15.

All experimental values were recorded in experiments when the movable element of the regulating device was installed under the action of the air flow and the balancing force of the dynamometer.

Previously, the error estimation was performed during experimental studies on the stand to study the properties of the regulating device. The total standard error of measurements is calculated using the formula:

\[
\Delta \sigma_m = \sqrt{\Delta \sigma_1^2 + \Delta \sigma_2^2 + \Delta \sigma_3^2 + \Delta \sigma_4^2 + \Delta \sigma_5^2 + \Delta \sigma_6^2 + \Delta \sigma_7^2 + \Delta \sigma_8^2}
\]  

where \( \Delta \sigma_1 \) – is the error pneumatic tube (±2 %); \( \Delta \sigma_2 \) – error differential manometer TESTO 510 (±2,5 %); \( \Delta \sigma_3 \) – error visual lifting of reading on the linear scale (to the naked eye is \( \Delta t = \pm 0.5 \text{ mm} \) (~1 %) and does not depend on the reference frame); \( \Delta \sigma_4 \) – error method of measurement according to the testimony pneumatic tube installed on the axis of the duct, (assumed to be 2 %); \( \Delta \sigma_5 \) – error of the measurement with a dynamometer, (accepted ~ 1 %); \( \Delta \sigma_6 \) – error due to possible leakage of the aerodynamic stand (the installation was sealed with plasticine and sticky tape, is estimated at 3 %); \( \Delta \sigma_7 \) – error, taking into account the inaccuracy of the geometric dimensions of the cross section of the
channel (for channel d=160 mm is estimated as 0.5 mm/160 mm=0.3 %); \( \Delta \sigma_8 \) – error removing static pressures with fittings (estimated at 1 %); \( \Delta \sigma_9 \) – error, taking into account the gap between the movable element and the guide on which it moves (estimated at 0.5 %).

\[
\Delta \sigma_n = \sqrt{2^2 + 2.5^2 + 1^2 + 2^2 + 1^2 + 3^2 + 0.3^2 + 1^2 + 0.5^2} = 5.2\% = 0.052
\]

The confidence probability of measurement \( \alpha \) was estimated. The confidence interval of measurements on the experimental stand should be comparable to the standard error of the installation, it is accepted with a margin of \( \Delta x=0.07 \). In the course of experiments, to reduce the error, it is assumed to perform \( n=3-4 \) similar measurements. The student coefficient values [24] are calculated using the formula:

\[
t_{\alpha} = \frac{\Delta x \cdot \sqrt{n}}{\Delta \sigma_n}
\]

- for 4 dimensions: \( t_{\alpha} = \frac{0.07 \cdot \sqrt{4}}{0.052} = 2.7 \)
- for 3 dimensions: \( t_{\alpha} = \frac{0.07 \cdot \sqrt{3}}{0.052} = 2.3 \)

Interpolating the table values of \( \alpha \), we find for \( n=4 \ \alpha=0.92 \), for \( n=3 \ \alpha=0.84 \). Such confidence probabilities correspond to the expected accuracy of the experimental studies performed. Accordingly, there would be a run of 3-4 of the same type of measurement in the experiments.

The local resistance coefficient \( \zeta \) and the parameter \( k \), which characterizes the forces on the mobile element of the control device, were determined using several similar measurements for each position of the mobile element. The results were then averaged. To reduce the errors associated with the vibration of the mobile element of the control device, two zones of movement of the mobile element were considered.

For the first zone with a cross-section length for air passage in the range from 55 to 15 mm, the length of such a cross-section was changed in increments of 5 mm. To obtain several experimental values, the flow rate was changed using the throttle-valve 4, and 4 similar measurements were performed.

For the second zone with a cross-section length for air passage from 15 to 9 mm, the step decreased to 2 mm. In the experiments performed, the minimum cross-section length for air passage was 9 mm. For each position of the movable element of the second zone, 3 identical measurements were performed at different air flow rates.

The experimental results were processed using the following formulas:

- axial air velocity in the channel \( V_{ax} \), m/s:

\[
V_{ax} = 1.29 \sqrt{P_T}
\]  \hspace{1cm} (8)

where \( P_T \) – is the differential pressure gauge reading corresponding to the differential of total and static pressure measured by a two-channel pneumatic tube;

- average speed in channel \( V_{av} \), m/s:

\[
V_{av} = 0.81 \cdot V_{ax}
\]  \hspace{1cm} (9)

- air flow in the channel \( L \), m³/s:

\[
L = V_{av} \cdot F
\]  \hspace{1cm} (10)

where \( F \) – is the cross-section area of the channel, for \( d=0.16 \) m, \( F=\pi \cdot d^2/4=0.0201 \) m².

- calculated \( V_{cp} \) value, dynamic pressure in the channel, \( P_{d} \), Pa:

\[
P_{d} = \frac{D \cdot V_{ax}^2}{2}
\]  \hspace{1cm} (11)
– coefficient of local resistance of the regulating device $\zeta$:

$$\zeta = \frac{P_f - R \cdot l_m}{P_d}$$  \hspace{1cm} (12)

where $P_f$ – the measured pressure drop between the fittings, Pa; $R$ – the resistivity, which was determined by reference tables, depending on the diameter and speed, Pa/m; $R \cdot l_m$ – the calculated pressure loss in the area between the fittings, Pa, where $l_m=1.5$ m;

The dimensionless characteristic of the force from the impact of the flow on the mobile element is the parameter $k$, which from the experimental data is found by the formula:

$$k = \frac{u_s / 102}{f_s \cdot P_f}$$  \hspace{1cm} (13)

where $u_s$ – is the force measured by the dynamometer, g; 102 – is the coefficient of conversion of forces measured in grams in N (Newtons); $f_s$ – sectional area of a round movable element at $r=0.055$ m, $f_s = \pi \cdot r^2 = 0.0094$ m$^2$.

3 Results and Discussion

Based on the processed results of the experimental study, experimental points are plotted on the graph of the dependence of the local resistance coefficient $\zeta$ on the ratio of the cross-section length for the passage of air $l$ to the diameter of the air duct $d \cdot l/d$ (Figure 5).

Similarly, experimental points are shown in the coordinates of the dependence of the parameter $k$, which characterizes the effect of the flow on the mobile element of the device on the values $l/d$ (Figure 6). The same graphs show the results of a numerical study to determine the values of $\zeta$ and $k$ obtained by the CFD method.

![Figure 5](link-to-image)

**Figure 5.** Graph of the dependence of the coefficient $\zeta$ on $l/d$: ● – averaged experimental values of the experimental study; □ – results of numerical research; solid line – a curve constructed according to the formula (14).

The experimental dependence $\zeta$ on $l/d$ in the studied range is approximated by a power function:

$$\zeta = 5.22 \cdot (l/d)^{0.85}$$  \hspace{1cm} (14)
Figure 6. Graph of the dependence of the coefficient $k$ on $l/d$: ● – averaged experimental values of the experimental study; □ – results of the numerical study; solid line – a curve constructed according to the formula (15).

The experimental dependence for the parameter $k$ is well approximated by the formula:

$$k = 1.05 \cdot (l/d)^{1.4}$$

(15)

As a confirmation of the correctness of the chosen method for determining the number of similar measurements, the actual spread of experimental points of values $\zeta$ (Figure 7a) and $k$ (Figure 7b) at different air flow rates for values $l/d=0.281$ and $l/d=0.125$.

Figure 7. Spread of experimental points. Solid line – waiting for the result; dashed lines – confidence interval boundaries; ▲●■♦ – individual experimental values of the experimental study at different air flow rates and the specified $l/d$ value.

Most test points are placed within the value confidence interval. For example, the figures show that 7 out of 8 most points are located within the confidence interval, i.e. for these experimental points, the confidence probability is $7/8=0.875$ coincides with the previously performed confidence assessment probabilities $\alpha=0.84–0.92$.

4 Conclusion
Based on numerical simulation, pressure distributions in axisymmetric flow sections in a channel with a cylindrical body and a diaphragm are calculated at a variable distance \( l \) from the cylindrical body to the diaphragm. Information about the pressure distribution allowed us to determine the coefficient of local resistance \( \zeta \) and the parameter \( k \), which characterizes the aerodynamic force acting on a cylindrical body depending on the relative size of \( l/d \).

The results of an experimental study of the aerodynamic characteristics of the same device, which is a model of the ventilation air flow regulator, performed on the stand, showed their satisfactory compliance with the results of the numerical study. Most of the experimental points lie within the confidence interval, which confirms the preliminary assessment of the accuracy of the study.

Thus, the main characteristics of \( \zeta(l/d) \) and \( k(l/d) \) are obtained. These results apply to all such devices with geometrically similar dimensions in a turbulent driving mode and can be used as a basis for calculating and designing a ventilation air flow controller for specific application conditions. Such devices are able to maintain an approximately constant set air consumption and thus ensure the rational use of energy, especially thermal energy in the cold season.

References

[1] Tong L, Gao J, Luo Z, Wu L, Zeng L, Liu G and Wang Y 2019 A novel flow-guide device for uniform exhaust in a central air exhaust ventilation system Building and Environment 149 pp 134-145 DOI: 10.1016/j.buildenv.2018.12.007
[2] Gao R, Wen S, Li A, Zhang H, Du W and Deng B 2019 A novel low-resistance damper for use within a ventilation and air conditioning system based on the control of energy dissipation Building and Environment 157 pp 205-214 DOI: 10.1016/j.buildenv.2019.04.041
[3] Broyda V A and Romanov V S 2019 Design characteristics of direct action airflow stabilizer with nonlinear stiffness of elastic element, included in the natural exhaust system Izvestiya KGASU 2(48) pp 182-189
[4] Zhang Q, Zhang M, Zhou Z and Wei S 2015 Numerical calculation of the tee local resistance coefficient Open Mechanical Engineering Journal 9(1) pp 876-881 DOI: 10.2174/1874155X01509010876
[5] Misaran M, Sing C and Adzrie M 2019 Study of wyee-tee duck design at various protrusion and guide vane location using CFD CFD Letters 11(11) pp 39-47 (2019)
[6] Ziganshin A M, Gimadieva G A and Batrova K E 2017 The pressure losses and the characteristics of the jet flowing through the middle lateral outlet Izvestiya KGASU 4(42) pp 257-265
[7] Taghipour R, Abdo P and Huynh B 2018 Effect of wind speed on ventilation flow through a two dimensional room fitted with a windcatcher Proceedings (IMECE) 7 pp 322-340 DOI: 10.1115/IMECE2018-88666
[8] Abdo P, Taghipour R and Huynh B 2019 Three dimensional simulation of ventilation flow through a solar windcatcher AJKFluids 2 pp 97-108 DOI: 10.1115/AJKFluids2019-5383
[9] Kunene T, Oliver G and Steyn J 2018 Determination of the head loss coefficient of closely spaced pipe bends SACAM pp 627-639
[10] Zaki A, Richards P and Sharma R 2019 Analysis of airflow inside a two-sided wind catcher building Journal of Wind Engineering and Industrial Aerodynamics 190 pp 71-82 DOI: 10.1016/j.jweia.2019.04.007
[11] Ferrucci M, Brocato M, Peron F and Cappelletti F 2015 Graphic and parametric tools for preliminary design stage of natural ventilation systems Building Simulation Applications pp 383-389
[12] Bangalee M, Lin S and Miau J 2016 Effects of porosity on wind driven natural cross ventilation of a full scale building Journal of the Chinese Society of Mechanical Engineers 37(4) pp 305-313 (2016)
[13] Keslerova R and Trdlicka D 2016 Numerical simulation of 3D flow of viscous and viscoelastic fluids in T-junction channel European Conference on Numerical Mathematics and Advanced Applications 112 pp 491-498 DOI: 10.1007/978-3-319-39929-4_47

[14] Gong Q, Yang J, Han K, Huang T, Li J and Zuo P 2016 Characteristic analysis on the flow and local resistance in large pipe tees Journal of Chinese Society of Power Engineering 36(9) pp 753-758

[15] Gao R, Li H, Li A, Liu K, Yu S and Deng B 2019 Applicability study of a deflector in ventilation and air conditioning duct tees based on an analysis of energy dissipation Journal of Wind Engineering and Industrial Aerodynamics 184 pp 256-264 DOI: 10.1016/j.jweia.2018.11.025

[16] Kosutova K, van Hooff T, Vanderwel C, Blocken B and Hensen J 2019 Cross-ventilation in a generic isolated building equipped with louvers: Wind-tunnel experiments and CFD simulations Building and Environment 154 pp 263-280 DOI: 10.1016/j.buildenv.2019.03.019

[17] Ameen A, Cehlin M, Larsson U and Karimipanah T 2019 Experimental investigation of the ventilation performance of different air distribution systems in an office environment-cooling mode Energies 12(7) pp 85-101 DOI: 10.3390/en12071354

[18] Chen H, Cai W and Chen C 2016 Fan-independent air balancing method based on computation model of air duct system Building and Environment 105 pp 295-306 DOI: 10.1016/j.buildenv.2016.06.008

[19] Cui C, Zhang X, Cai W and Jing G 2019 A gradient-based adaptive balancing method for dedicated outdoor air system Building and Environment 151 pp 15-29 DOI: 10.1016/j.buildenv.2019.01.015

[20] Cheng F, Cui C, Zhang X, Cai W, Ge Y, Gao W and Mao T 2019 A robust air balancing method for dedicated outdoor air system Energy and Buildings 202 pp 109-122 DOI: 10.1016/j.enbuild.2019.109380

[21] Jing G, Cai W, Zhang X, Cui C, Yin X and Xian H 2019 Modeling air balancing and optimal pressure set-point selection for the ventilation system with minimized energy consumption Applied Energy 236 pp 574-589 DOI: 10.1016/j.apenergy.2018.12.026

[22] Zuo H, Li X and Tu J 2015 An energy saving ventilation strategy for short-term occupied rooms based on the time-dependent concentration of CO2 International Journal of Ventilation 14(1) pp 39-52 DOI: 10.1080/14733315.2015.11684068

[23] Kobayashi T, Sugita K, Umemiya N, Kishimoto T and Sandberg M 2017 Numerical investigation and accuracy verification of indoor environment for an impinging jet ventilated room using computational fluid dynamics Building and Environment 115 pp 251-268 DOI: 10.1016/j.buildenv.2017.01.022

[24] Zhao W and Zhang R 2015 Variable selection of varying dispersion student-t regression models Journal of Systems Science and Complexity 28(4) pp 961-977 DOI: 10.1007/s11424-014-2223-9