

## O. Gumen

Professor  
National Technical University of Ukraine  
'Kyiv Polytechnic Institute' of Kyiv  
Department of descriptive geometry,  
engineering and computer graphics

## V. Dovhaliuk

Professor  
Kyiv National University of  
Construction and Architecture of Kyiv  
Department of Heat Gas Supply  
and Ventilation

## V. Mileikovskiy

Associate Professor  
University of Nis  
Kyiv National University of  
Construction and Architecture of Kyiv  
Department of Heat Gas Supply  
and Ventilation

## O. Lebedieva

Senior Lecturer  
National Technical University of Ukraine  
'Kyiv Polytechnic Institute' of Kyiv  
Department of descriptive geometry,  
engineering and computer graphics

## V. Dziubenko

Associate Professor  
University of Nis  
Kyiv National University of  
Construction and Architecture of Kyiv  
Department of Labour Protection  
and Environment

# Geometric Analysis of Turbulent Macrostructure in Jets Laid on Flat Surfaces for Turbulence Intensity Calculation

*European Norms for ventilation contain turbulence intensity requirements in rooms. One of the determining factors is turbulence intensity in ventilation jets. We found an approach for geometric analysis of turbulent macrostructure for subsonic flows with large-scale vorticity i.e. ventilation jets and boundary layers between flows. This approach requires building of simplified turbulent macrostructure chart and performing geometrical analysis of it. In previous works, using the approach we analytically found averaged characteristic of free jets, jets in flows and jets laid on different shape surfaces without requirements of any experimental data. The results of geometrical analysis of heat transfer between flows are used in Ukrainian norms. In this work we found turbulence intensity of wall jets on flat surfaces such as room walls or ceiling without any experimental values. The results are coincide with known experimental data and may be used in flow calculation in rooms.*

**Keywords:** turbulence intensity; turbulent macrostructure; jet; wall jet; computational fluid dynamic.

## 1. INTRODUCTION

Human thermal comfort is human body ability to dissipate all heat energy generated by metabolism without stress on human heat regulation system. Heat exchange between the body and the internal room air is dependent on five microclimate parameters, covered by European Norms: air temperature, relative humidity, air velocity, average surface temperature (radiant temperature) and turbulence intensity.

During harmonization with European Norms Ukraine accepted turbulence intensity standardization [1]. Most of air distributors datasheets contain turbulence intensity charts but they do not consider interaction effects between jets, other flows and obstacles. Theories of turbulent flows are not developed well enough to easily calculate the turbulence intensity. The most precise turbulence model [2] is DNS, which based on direct numerical solution of Navier-Stokes equations (obtained for laminar flows). For turbulent flows the equations loose stability but remain valid if cell size of calculation mesh is less than the smallest possible eddy. Solutions of the equations on many time steps may be averaged. The solution of small box  $47\delta \times 18\delta \times 9.6\delta$ , equipped by thin inlet air slot with width  $\delta$ , require 22000 processor hours. Such

calculations can be performed only on large and very expensive computer clusters. To decrease the computing resource consumption the mesh may be rougher using the special equations for lowest eddies (subgrid vorticity). But for significant decrease of mesh complexity another concept of computational fluid dynamics (CFD) simulation is used [3]. The Reynolds-averaged [3] Navier-Stokes equations (or Reynolds equations) use Reynolds-stresses  $\rho u'_i u'_j$  ( $\rho$  – density;  $u'_i$  and  $u'_j$  – pulsation velocity components along axes  $i$  and  $j$  (may be the same –  $i=j$  – or not –  $i \neq j$ )). They are dependent on velocity pulsation components, not on physical properties. The additional equations required for the equation system closure. Turbulent (eddy) viscosity  $\epsilon_m$  may be used instead of the Reynolds stresses. It is a correction for physical kinematic viscosity coefficient  $\nu$  ( $\nu + \epsilon_m$ ). Both values may be treated as fictitious because they are characteristics of turbulent pulsations and their energy, not a physical properties. They theoretically can be found directly from Navier-Stokes equations (i.e. by DNS). The problem that we have no effective and low resource consuming mathematical apparatus for direct solution of highly unstable equations. Thus we need additional redundant assumptions.

There are early classical [3] concepts of turbulent mixing length by L. Prandtl, eddy viscosity concept by J. Boussinesq etc. CFD simulation based on partial differential equation has been founded by A. Kolmogorov and J. Rotta. The discovered principles are actual for modern CFD approaches.

Received: October 2016, Accepted: October 2016

Correspondence to: Viktor Mileikovskiy

Vul. Ivana Mazepy, 14, kv. 11-A,

Kyiv, 01010, Ukraine

E-mail: mileikovskiy@gmail.com

Zero-equation models [3] describe the turbulence viscosity by algebraic equations. Cebeci-Smith, Baldwin-Lomax models are examples of zero-equation approaches. More universal are one-equation models with single additional differential equation (transport equation). But the models with two transport equations (two-equation models) are more universal, thus the most popular in commercial CFD software. The most widely used models [3] are k- $\epsilon$  model (the most popular) by W. Jones and B. Launder, k- $\omega$  model by D. Wilcox and SST model by F. R. Menter, using k- $\omega$  model for outer regions and k- $\epsilon$  for inner regions.

Large vorticity have much more amount of energy than small one. The idea [4] is that if large-scale vorticity is in a flow the small-scale vorticity is not important and can be neglected in simulation by low-pass filtering. This model is known as Large Eddy Simulation (LES) and it is very useful for jets, mixing layers, separation flows etc. Hybrid of Reynolds-averaged Navier-Stokes equation solution and LES simulation of separation flows is called Detached Eddy Simulation (DES). We will finish this very brief excursion of models. These and other models are well discussed in the literature such as [3-7]. CFD is the most useful method of solving the most of tasks.

The first disadvantage of this approach is lot of experimental coefficients (at least five for k- $\epsilon$  model). There is no prove that their most commonly used values are universal and useful for all possible boundary and initial conditions. The most of simulation results such as [8] are successful. Nevertheless, the simulation [9] of air diffusers with multiple slots, tangential to cylindrical surface, shows that formed jets interact if number of slots is four. The experimental research [9] show that the number must be at least five. Thus, experimental validation of simulation results for complex problems is strongly recommended.

The second disadvantage is that simulation process is principally same as physical experiments except it does not require actual laboratory experiments. For single installation of already designed inflows and outflows, CFD process is very quick because we need only few simulations. But there are more complex tasks i.e. optimization or obtaining of some dependency of a response function in multidimensional factor space. The solution is performing design of experiments and realization of the design matrix in "virtual laboratory" – CFD software. It is much faster and cheaper than laboratory tests but require a lot of time for 3D model (mesh) building and calculations using high-cost hardware and software. There is no possibility of direct optimization or engineering equation construction from the CFD model equations. Therefore, the turbulent flows theory development is good idea for air distribution design simplification.

The third disadvantage is using of fictitious values such as turbulent stress, turbulent viscosity, dynamic velocity, turbulent Prandtl number etc. The physical meaning of the values is not obvious as pressure, velocity or physical viscosity coefficients (kinematic and dynamic). Thus, there are additional difficulties learning and understanding the turbulence models.

A. Tkachuk, the professor, chair of Heat Gas Supply and Ventilation Department of Kyiv National University of Construction and Architecture [10] has developed new theory of turbulent boundary layers using the singularity method. Turbulent flow is regarded as a stream of ideal liquid with small vortices as 'singularities'. Turbulent boundary layers is simplified as vortex films of adjoining vortex cords. Using the Kelvin-Stokes theorem this theory describes the influence of vortices directly avoiding additional values with unapparent physical meaning i.e. turbulence viscosity, mixing length or turbulent Prandtl number. The theory has been originally developed only for averaged flow and it did not cover turbulence parameters.

We propose a continuation of A. Tkachuk's researches. Based on different visual researches [11-15] and many other, a simplified approach is offered for jet flows and other flows with large-scale vorticity, which simplifies such flows as a group of adjoining round large-scale vortices (puffs). It can give averaged parameters of the flows based on geometrical and kinematic analysis only avoiding integration in the most of cases.

In previous works, using the approach we analytically found averaged characteristic of free jets, jets in flows and jets laid on different shape surfaces without requirements of any experimental data. Results for heat transfer between flows in lower feed pipe of radiator nodes at closed thermostatic radiator valves (back flow effect) in one-pipe heating systems are used in Ukrainian norm [16].

The approach can describe not only the average parameters but also the low-frequency parameter changes (turbulent pulsations) caused by the puffs. In this work the turbulent parameters of a jet laid on a flat wall (flat wall jet) will be calculated. Using Tolmien source concept [17] the jet will be considered as discharged from infinitely small slot located at the pole  $P$  (intersection point of the jet boundaries at large distance from the real slot) of the jet. As the puffs have incomparably more energy than small-scale vorticity, they almost determine flow characteristics including turbulence intensity.

An abscissa  $x$  is usually aligned with the wall. For jets with maximum average (by time) velocity  $u_m$  in a section, local (at a point of the same section) instantaneous (non-averaged)  $x$ -velocity  $u_x$ , local average (by the time) velocity  $\bar{u}_x$  and local pulsation velocity  $u_x^I$ , the turbulence intensity

$$\varepsilon = \frac{u_x^I}{u_m}. \quad (1)$$

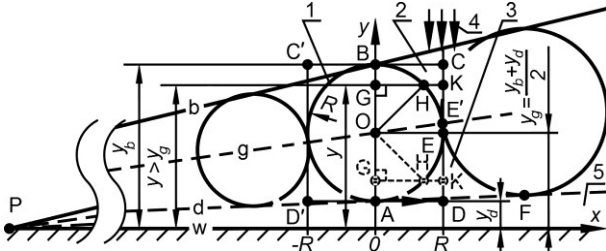
The pulsation velocity is most commonly defined as root mean square (RMS) of the velocity  $u_x$ :

$$u_x^I = \sqrt{(u_x - \bar{u}_x)^2}, \quad (2)$$

where bar means averaging by time.

## 2. MAIN CONCEPTS

Let us consider (Figure 1) a flat jet from infinitely small slot  $P$  near to a flat wall  $w$  as a puff sheet. Let us choose a puff 1. Between the puffs there are interpuff layers with external parts 2 and internal parts 3. The ambient air (or gas or liquid) inflows 3 to the jet in the normal direction to the wall  $w$ . The  $x$ -axis is coincident to the wall  $w$  in the jet direction and the  $y$ -axis runs from the wall to the ambience in the section  $AB$  crossing the puff 1 centre  $O$ .



**Figure 1. Simplified macrostructure chart of wall jets: 1 – puff; 2 – external part of interpuff layer; 3 – internal part of interpuff layer; 4 – inflow from the ambience; 5 – wall boundary layer; line  $b$  – free boundary; line  $g$  – puff centres locus; line  $d$  – division line between puff jet boundary layer; line  $w$  – the wall**

The jet has [17] two layers: the wall boundary layer (between the lines  $w$  and  $d$ ) with a small-scale turbulent structure (out of scope) and the jet boundary layer with puffs (between lines  $d$  and  $b$ ). For the Tolmien source (as at the enough distance from the real jet beginning) the lines  $b$  (the free boundary),  $g$  (the puff centres locus) and  $d$  (the division line between puff jet and wall boundary layers) are straight. In this context the line  $d$  is not the maximum velocity line – the locus of points with maximum velocity  $u_m$  in all jet sections.

The puffs may form, deform and destroy. But for the most common (averaged) jet state the puffs may be considered as adjoining circular cylindrical vortex cords, rolling on the free jet boundary as it is considered by A. Tkachuk for turbulent boundary layers with small macrostructure [10]. On the Figure 1 there are the puff 1 radius  $R$ , high  $y_b$  of the section  $AB$ , high  $y_d$  of the wall boundary layer in the section and the distance  $y_g$  from the wall  $w$  to the centre  $O$  of the puff 1.

It is possible to neglect growth of the puff 1 during its full movement through the section  $AB$ . The right endpoint  $E$  of horizontal diameter of the puff 1 and the puffs touch point  $E'$  are very close. Thus we can replace time averaging by averaging along  $x$  from minus  $R$  to  $R$  inside the rectangle  $C'DD'$ . The Figure 1 shows that at one half of the range the required averaged value may be underestimated and at another half this value may be overestimated. Therefore, the average value on the full range may be very close to the true value. Also neglecting of the puff growth cause reflection symmetry of the  $x$ -velocity field respect to the  $y$ -axis inside the range. It is enough to use the range along the  $x$ -axis from  $x=0$  to  $x=R$  inside the rectangle  $ABCD$ .

By the Euler formulas [18]  $x$ -velocity  $u_{x,p}$  of the rotating puff is linearly dependent on  $y$ -coordinate normal to the wall and independent on  $x$ -coordinate. The puff 1 velocity at the point  $A$  is denoted as  $u_p$ . The

inflow in the external part 2 of the interpuff layer has no  $x$ -velocity. In the internal interpuff layer 3 it is only possible to approximate  $x$ -velocity. At the points  $A$  and  $D$  velocity is near to  $u_p$ . Velocity above the point  $D$  may reach  $u_p$ . At the point  $E$  it is equal to translation velocity of the puff 1  $u_p/2$ . The simplest approximation is usable – averaged constant  $u \approx (3/4)u_p$ .

In the puff 1 linear  $x$ -velocity dependency is given by zero value at the instantaneous rotation point (axis)  $B$  and peripheral velocity  $u_p$  at the opposite point  $A$ :

$$u_{x,p} = u_p \frac{y_b - y}{y_b - y_d} = u_p \left( 1 - \frac{y - y_d}{y_b - y_d} \right). \quad (3)$$

The following equations for interpuff  $x$ -velocity is helpful to avoid additional calculations:

$$u_{x,I} = u_p P \left( \frac{y - y_d}{y_b - y_d} \right); \quad (4)$$

$$P \left( \frac{y - y_d}{y_b - y_d} \right) = \begin{cases} 0 & \text{if } y \geq y_g = \frac{y_b + y_d}{2}; \\ \frac{3}{4} & \text{if } y < y_g = \frac{y_b + y_d}{2}. \end{cases} \quad (5)$$

## 3. GEOMETRIC ANALYSIS OF THE SIMPLIFIED MACROSTRUCTURE CHART

Let us choose some  $y$  value. In the rectangle  $ABCD$  at the  $y$  level on the Figure 1 there is the line  $GK$  that intersects the puff 1 at the point  $H$ . There are three cases:  $y > y_g$  (solid line);  $y = y_g$  (cause  $O \equiv G$  and  $E \equiv H \equiv K$  coincidences, not shown on the Figure 1);  $y < y_g$  (short dashed line).

The diameter of the puff 1  $AB$  has length

$$|AB| = y_b - y_d. \quad (6)$$

Length of the lines  $GH$  and  $HK$  –  $|GH|$  and  $|HK|$  are necessary. It can be found from the right-angled triangle  $OGH$  with the right angle  $G$ . Length  $|OH|$  of the line  $OH$  is equal to length  $|GK|$  of the line  $GK$  and equal to radius  $R$  of the puff 1. Using the equation (6):

$$|OH| = |GK| = R = \frac{|AB|}{2} = \frac{y_b - y_d}{2}, \quad (7)$$

where  $|AB|$  is length of the diameter  $AB$  of the puff 1. Length of the line  $GO$  is

- $|GO| = y - y_g$ , if  $y > y_g$ ;
- $|GO| = 0$ , if  $y = y_g$ ;
- $|GO| = -(y - y_g)$ , if  $y < y_g$ .

The most common equation, that covers all cases, is

$$|GO| = |y - y_g| = \frac{|2y - y_b - y_d|}{2}. \quad (8)$$

Therefore, using the equations (7) and (8)

$$|GH| = \sqrt{|OH|^2 - |GO|^2} =$$

$$\begin{aligned}
&= \sqrt{\left(\frac{y_b - y_d}{2}\right)^2 - \left(\frac{2y - y_b - y_d}{2}\right)^2} = \\
&= \frac{1}{2} \sqrt{(y_b - y_d)^2 - (2y - y_b - y_d)^2}. \quad (9)
\end{aligned}$$

As length of the line  $GK$  is  $|GK| = R$ , length of the line  $HK$ , using the equations (7) and (9), is

$$\begin{aligned}
|HK| &= |GK| - |GH| = \\
&= \frac{y_b - y_d - \sqrt{(y_b - y_d)^2 - (2y - y_b - y_d)^2}}{2}. \quad (10)
\end{aligned}$$

The equations (9) and (10) are valid for any case. After simple transformations:

$$|GH| = (y_b - y_d) \sqrt{\left(\frac{y - y_d}{y_b - y_d}\right) \left(1 - \frac{y - y_d}{y_b - y_d}\right)}; \quad (11)$$

$$\begin{aligned}
|HK| &= (y_b - y_d) \times \\
&\times \left( \frac{1}{2} - \sqrt{\left(\frac{y - y_d}{y_b - y_d}\right) \left(1 - \frac{y - y_d}{y_b - y_d}\right)} \right). \quad (12)
\end{aligned}$$

Ordinate  $y_g$  of the puff 1 centre  $O$  is simple mean of ordinates of the diameter endpoints  $A$  and  $B$ :

$$y_g = \frac{y_b + y_d}{2}. \quad (13)$$

#### 4. AVERAGING OF VELOCITY AND ITS DEVIATION

Averaging by the line  $GK$  of a value  $v$ , that is a constant  $v_p$  in the puff 1 and another constant  $v_l$  in the interpuff layer, may be performed by the following simple formula using the equations (7), (11) and (12):

$$\begin{aligned}
\bar{v} &= \frac{v_p |GH| + v_l |HK|}{|GK|} = \\
&= 2v_p \sqrt{\left(\frac{y - y_d}{y_b - y_d}\right) \left(1 - \frac{y - y_d}{y_b - y_d}\right)} + \\
&+ v_l \left( 1 - 2 \sqrt{\left(\frac{y - y_d}{y_b - y_d}\right) \left(1 - \frac{y - y_d}{y_b - y_d}\right)} \right). \quad (14)
\end{aligned}$$

First, it is necessary to calculate average x-velocity using  $v = u_x$  by the equations (3), (4) and (14):

$$\begin{aligned}
\bar{u}_x &= u_p \left( 1 - \frac{y - y_d}{y_b - y_d} \right) \sqrt{\left(\frac{y - y_d}{y_b - y_d}\right) \left(1 - \frac{y - y_d}{y_b - y_d}\right)} + \\
&+ u_p P \left( \frac{y - y_d}{y_b - y_d} \right) \times \\
&\times \left( \frac{1}{2} - \sqrt{\left(\frac{y - y_d}{y_b - y_d}\right) \left(1 - \frac{y - y_d}{y_b - y_d}\right)} \right). \quad (15)
\end{aligned}$$

After elementary simplifications of the equation (15) using different brackets only for referencing

$$\begin{aligned}
\bar{u}_x &= u_p \left\{ 2 \left[ 1 - \frac{y - y_d}{y_b - y_d} - P \left( \frac{y - y_d}{y_b - y_d} \right) \right] \times \right. \\
&\times \left. \sqrt{\left(\frac{y - y_d}{y_b - y_d}\right) \left(1 - \frac{y - y_d}{y_b - y_d}\right)} + P \left( \frac{y - y_d}{y_b - y_d} \right) \right\}. \quad (16)
\end{aligned}$$

The equation (5) can be transformed to the following:

$$P \left( \frac{y - y_d}{y_b - y_d} \right) = \begin{cases} 0 & \text{if } \frac{y - y_d}{y_b - y_d} \geq \frac{1}{2}; \\ \frac{3}{4} & \text{if } \frac{y - y_d}{y_b - y_d} < \frac{1}{2}. \end{cases} \quad (17)$$

The maximum of the velocity profile (16) and (17) may be found using only school-level mathematics by plotting a chart of the multiplier in the braces dependent on the simplex in the round brackets or by derivative analysis (by hand or using computer algebra system i.e. Maxima). The last option gives the following precise value of the maximum ordinate  $y_m$  and the maximum velocity  $u_m$ :

$$\frac{y_m - y_d}{y_b - y_d} = \frac{7 - \sqrt{33}}{16} = 0.07846...; \quad (18)$$

$$u_m = u_p \frac{\sqrt{414 - 2\sqrt{33^3} + 48}}{64} = (0.84225...)u_p. \quad (19)$$

The first option gives an approximation of the results (18, 19). Let us put  $v = (u_x - \bar{u}_x)^2$  to the equation (14) using the equations (3), (4) and (16) and after that calculate the square root accordingly to the equation (2). The following form of RMS will be obtained:

$$\begin{aligned}
u_x^I &= \frac{1}{2} u_p \left[ 1 - \frac{y - y_d}{y_b - y_d} - P \left( \frac{y - y_d}{y_b - y_d} \right) \right] \times \\
&\times \sqrt{1 - \left( 1 - 4 \sqrt{\left(\frac{y - y_d}{y_b - y_d}\right) \left(1 - \frac{y - y_d}{y_b - y_d}\right)} \right)^2}. \quad (20)
\end{aligned}$$

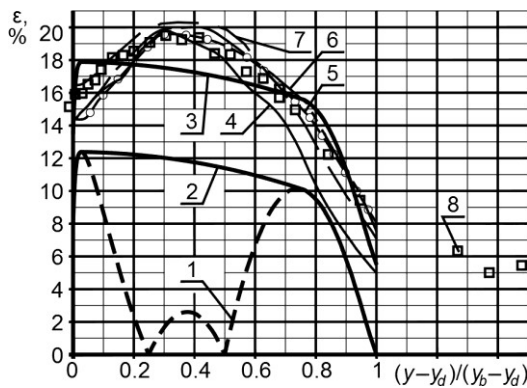
By the equations (1), (18), (19) and (20) turbulence intensity sought is

$$\begin{aligned}
\varepsilon &= \frac{32}{\sqrt{414 - 2\sqrt{33^3} + 48}} \times \\
&\times \left[ 1 - \frac{y - y_d}{y_b - y_d} - P \left( \frac{y - y_d}{y_b - y_d} \right) \right] \times \\
&\times \sqrt{1 - \left( 1 - 4 \sqrt{\left(\frac{y - y_d}{y_b - y_d}\right) \left(1 - \frac{y - y_d}{y_b - y_d}\right)} \right)^2} \approx
\end{aligned}$$

$$\approx 0.6 \left| 1 - \frac{y-y_d}{y_b-y_d} - P \left( \frac{y-y_d}{y_b-y_d} \right) \right| \times \sqrt{1 - \left( 1 - 4 \sqrt{\left( \frac{y-y_d}{y_b-y_d} \right) \left( 1 - \frac{y-y_d}{y_b-y_d} \right)} \right)^2} \quad (21)$$

## 5. COMPARISON WITH EXPERIMENTAL DATA

The results 1 of calculations by the equations (17) and (21) on the Figure 2 show obvious underestimation of the turbulence intensity, especially, at the middle of the jet. It is predictable because near to the puff touching point the x-velocity change is very small. However, the Tkachuk's theory [10] require simulation of the tangential velocity rupture by very intensive secondary medium-scale vorticity, produced by the puffs. It is not so easy but the peaks can be connected by a straight line or a convex curve (Figure 2). Physical meaning requires smooth turbulence intensity profile. Therefore, tangency conditions are used in Figure 2 in end points of the connection line.



**Figure 2. Turbulence intensity: 1 – results by the equation (21); 2 – the results with connected peaks using tangency conditions; 3 – the results with added correction + 5,5 % for turbulent vortices in inflow; 4 – DNS simulation results in [15] at distance from an inlet slot  $x$  relative to width  $\delta$  of the slot  $\delta/h = 40$ ; 5 – the same but  $\delta/h = 30$ ; 6 –  $\delta/h = 20$ ; 7 –  $\delta/h = 15$ ; 8 – experimental data of J. G. Eriksson, R. I. Karlsson, and J. Persson at  $\delta/h = 70$  [15]. All simulation and experimental data 4-8 is rescaled to  $u_m$**

The advantage of the proposed approach is the possibility of maximum turbulent intensity prediction in a section. The maximum is  $\varepsilon = 0.124$  or 12.4 %. Let us compare it with known experimental data. One of the problems that experimental data for wall jets in some countries [19-21] is usually presented in different way that is (1) for free jets [5]:

$$\varepsilon_\tau = \frac{u_x}{u_\tau}, \quad (22)$$

where  $u_\tau$  is friction velocity [19] or dynamic velocity [10]. It is not actual velocity but a parameter with the corresponding unit dependent on shear stress on the wall  $\tau_0$  and density  $\rho$ :

$$u_\tau = \sqrt{\frac{\tau_0}{\rho}}. \quad (23)$$

The value of  $\tau_0$  by the Newton law [10] is dependent on the velocity gradient at the wall multiplied by the dynamic viscosity  $\eta$ :

$$\tau_0 = \mu \left. \frac{du}{dy} \right|_{y=0}. \quad (24)$$

As the velocity (not its derivative) can be measured only at finite (may be very small) distance from the wall with some uncertainty, the derivative in the equation (24) may be approximated with significant deviation or calculated by any theory that always have some simplifications. In the work [19] on the page 8 at kinematic viscosity  $\nu$

$$\frac{\tau_0}{\rho u_m^2} = 0.0315 \left( \frac{u_m y_m}{\nu} \right)^{-0.182}; \quad (25)$$

$$\frac{u_m y_m}{\nu} \approx 10^4. \quad (26)$$

Using the equations (23), (24), (25) and (26)

$$u_\tau = 0.054 u_m. \quad (27)$$

At the Figure 11 of the work [19]  $\varepsilon_\tau = 3.3...3.5$ . By the equations (1), (22) and (26)  $\varepsilon = (3.3...3.5) \cdot 0.054 = 0.18...0.19$ . On Figure 4.3 of the work [21] with close conditions and more experimental points  $\varepsilon_\tau = 2.9...3.7$  excluding some points. The Figure 12 a of the work [2] shows the maximum value by DNS simulation  $\varepsilon_\tau = 2.5$  at the distance  $x/h$  equal to 25, 30 and 40, where  $h$  is slot widths. The Reynolds number at the slot  $u_0 h / \nu = 2000$ . Initial Mach number is 0.5. Figure 7 of the work [2] shows the corresponding friction velocity, related to the initial velocity,  $u_\tau / u_0 = 0.0413, 0.0402, 0.0363$ . Therefore  $\varepsilon = (2.9...3.7) \cdot 0.054 = 0.16...0.20$ . So, the difference between the result of the work (0.124) and experimental data is 0.036...0.076 or 3.6...7.6 %. The experimental data [15] show (Figure 2) that around the jet the turbulence intensity is nonzero and it is around 5...5.5 %. Maybe, it is caused by small-scale vorticity in inflow 4 (Figure 1). If this air (or liquid/gas) fully consumed by the jet the vortices may also be consumed. If the results are corrected by moving the obtained curve up by 5.5 %, we will obtain excellent coincidence with experimental data (in the central region the deviation is 2...3 % that is the same order as experimental uncertainty).

There is a very important advantage of this approach: it uses elementary geometry and kinematic knowledges and does not use hard to understand fictitious quantities with vague physical meaning, so we can use it to explain the very difficult aerodynamic task for wide range of people.

## 6. CONCLUSION

The proposed simplified approach give us a possibility to estimate the turbulence intensity in wall jets caused by the large-scale vorticity. Because the large-scale vorticity have the main influence, the deviation is 3.6...7.6 %. This deviation is the estimation of small-scale turbulence influence. The results can be corrected by adding the near to boundary turbulence intensity – 5.5 %. The deviation is up to 2...3 %. The advantage of this approach is its simplicity and absence of the additional values with indistinct meaning such as turbulent viscosity, mixing length etc. The future work will describe turbulence intensity of wall jets on walls with different curvature.

## ACKNOWLEDGMENT

The authors thanks for Andrei Tkachuk (1928-2002), the professor, chair of Heat Gas Supply and Ventilation department of Kyiv National University of Construction and Architecture for foundations of this research direction.

## REFERENCES

- [1] Strelchuk, O. and Sizov, O., DBN V.2.5-67:2013. *Opalennia, ventyliatsiia ta kondytsionuvannia. Vydannia ofciine*, Ukrarhbudinform, Kyiv, 2013 (in Ukrainian).
- [2] Ahlman, D., Brethouwer, G. and Johansson, A.V.: Direct Numerical Simulation of a Plane Turbulent Wall-Jet Including Scalar Mixing – Physics of Fluids, vol. 19 No. 6, pp. 065102-1 - 065102-13, 2007.
- [3] Cebeci, T., Shao, J. P., Kafyke F. and Laurendeau E.: *Computational Fluid Dynamics for Engineers*, Springer, Horizons Publishing, Long Beach, Heidelberg, 2005.
- [4] Volkov, K. N. and Emelianov, V. N.: *Modelirovaniie krupnykh vikhrei v raschetah turbulentnykh techenii*. Uchebnoe posobiie, Sankt-Petersberg, 2005.
- [5] Versteeg, H. K. and Malalasekera, W.: *An Introduction to Computational Fluid Dynamics. The Finite Volume Method*, Longman Scientific & Technical, Harlow, 1995.
- [6] Chung, T. J.: *Computational Fluid Dynamics*, Cambridge University Press, Cambridge, 2002.
- [7] Tannehill, J. C., Anderson D. A. and Pletcher R. H.: *Computational Fluid Mechanics and Heat Transfer*, Taylor & Francis, Washington, 1997.
- [8] Thool, S. B. and Sinha, S. L.: Simulation of Room Airflow using CFD and Validation with Experimental Results. *International Journal of Engineering Science and Technology*, No. 6(5), pp. 192–202, 2014.
- [9] Dovhaliuk, V. B. and Mileikovskiy V. O.: Formuvannia strumyn pry tanhentsialnykh vypuskakh povitria z povitrorozpodilnyka – Nova Tema, No. 2, pp. 22-24, 2007 (in Ukrainian).
- [10] Dovhaliuk, V.B.: *Aerodynamika ventyliatsii*. Navchalnyi posibnyk, Ukrheliotekh, Kyiv, 2015 (in Ukrainian).
- [11] Van Hooff T. et al.: PIV Measurements of a Plane Wall Jet in a Confined Space at Traditional Slot Reynolds Numbers, *Exp Fluids*, No. 53(2), pp. 499–517, 2012.
- [12] Juodis, E.S. and Motuziene, V.: *Vedinimo aerodinamika*. Vadovelis. ‘Technika’, Vilnius, 2014 (in Lithuanian).
- [13] Papanicolaou, P.N. and Gharib M.: Growth of a Round Jet, Under Local Reynolds Number Gradients, in: *Proceedings of the NATO Advanced Research Workshop on Recent Research Advances in the Fluid Mechanics of Turbulent Jets and Plumes*. Viana do Castelo, Portugal. Series E: Applied Sciences, Vol. 255. pp 177–190, 1994.
- [14] Schumamaker, S.A. and Driscoll J.F.: Mixing Lengths of Reacting and Nonreacting Coaxial Injectors in a Laboratory Rocket Combustor. 44th AIAA/ ASME/SAI/ASEE Joint Propulsion Conference and Exhibit. 21-23 July 2008, Hartford, CT. pp. 177–190.
- [15] Zhukovskiy, S.S. and Labai V.I.: *Aerodynamika Ventyliatsii*. Navchalnyi posibnyk, Vidavnistvo Natsionalnogo universitetu ‘Lvivska politekhnika’, Lviv, 2003 (in Ukrainian).
- [16] Babicheva, P., Voinalovich, I., Galinskyi, O., Gutnichenko, T., Ivanenko, V. and Maksimov, A. *DSTU-N B V.3.2-3:2014. Guidelines for the Implementation of Thermomodernisation of Residential Building*, DP “UkrArkhBudInform”, 2014
- [17] Abramovich, G.N.: *Teoriia turbulentnykh strui*, Ekolit, Moscow, 2011 (in Russian).
- [18] Drong, V.I. et al.: *Kurs teoreticheskoi mekhaniki*. Uchebnyk dlia vuzov. 3-e izd., MGTU im. N.E. Bauman, Moscow, 2005 (in Russian).
- [19] Bradshaw, P. and Gee, M.T.: *Turbulent Wall Jets with and without an External Stream*. Reports and Memoranda No. 3252\*. June, 1960. Her Majesty’s Stationary Office, London, Great Britain, 1962.
- [20] Zhujun, T. et al.: Reynolds Stress Measurements in a Plane Turbulent Wall Jet on a Smooth Surface, in: *Proceedings of the International Symposium on Turbulence and Shear Flow Phenomena (TSFP-9)*, 30.06-03.07. 2015, Melburn, pp. 281–305.
- [21] Eriksson, J.: *Experimental studies of the plane turbulent wall jet*. Technical reports from Royal Institute of Technology Department of Mechanics FaxenLaboratoriet SE-100 44. Stockholm, Sweden, 2003.