FRI-9.3-1-THPE-05

NUMERICAL MODELING THE INTERACTION OF TURBULENT JET WITH SUCTION OPENING WITH PRESENCE OF AN ANGLE BETWEEN SECTIONS

PhD student Ivan Denev, Eng.
Department of Hydroaerodynamics and Hydraulic machines
Phone: +359893690127
E-mail: ivan_denev.eng@abv.bg

Chief assistant professor Petko Tsankov, PhD
Technical university of Sofia, Bulgaria, Branch Sliven
Tel.: +359895581011
E-mail: ptsankov@abv.bg

Prof. Ivan Antonov, DSc
Department of Hydroaerodynamics and Hydraulic machines
Technical university of Sofia, Bulgaria
Tel.: +359892231033
E-mail: mfantonov@abv.bg

Abstract: *This paper is about using computational fluid dynamics for obtaining a solution concerning interaction of a turbulent jet with suction opening with presence of an angle between the sections. The accuracy of a modern calculation method give a reason to be used in the design of local exhaust ventilation. This option with the presence of an angle between the sections is interesting for modeling because most of the technological processes in different industries are related to the separation of heated vapor and fine particles or such with greater specific weight than air whose capture requires the use of angle between the injection jet and the suction opening.*

Keywords: *turbulent jet, CFD modeling, suction opening, pollutants*

INTRODUCTION

Different industrial manufactures except those of goods for the society and value added, are accompanied with release of harmful substances. It's about significant amounts of heat, moisture, vapor and waste gases, dust and aerosols. To be ensured their effective removal with purpose to not allow the worsened parameters of microclimate to threaten the health of people, working in the production facilities ventilation installations are being built.

Forming zones with concentrated pollutant separation in the production facilities around certain workplaces or technological processes requires local suction ventilation systems (Olander L., Industrial ventilation design guidebook, Chapter 10 Local Ventilation).

They are especially suited for applications where in a working room several processes are going on accompanied by the release of different types of harmful substances. Usually the separated pollutants in production facilities are classified in tree groups - powder aerosols, smoke, where the dispersed phase represents solid particles; mist with liquid particles as well as vapor and gas. Typical for the particles with size smaller than 40 mm for example, vapor and gas is that they follow the air flow - they are orientated in the direction of its current lines. So it is good suction ducts, intended to capture them, to be designed so that the speed of the intake air corresponds to the speed of the contaminated air flow. If the size of the particles is bigger than 40 mm, at a high velocity of polluted airflow it is observed their detachment from the air flow. The particles are distracted in the working area following their own trajectory, in case there are no suitable enclosures. Effective capture of such an aerosol is achieved by placing a suction duct normally on the particle trajectory. This requires the use of suction ducts located at an angle to the turbulent jet carrying pollutants. The purpose of this work is to numerically simulate the flow in the on-board suction duct (Penev S. (2001). Promishlena ventilatsiya I obezprashavane, Sofiya (***Оригинално заглавие:*** *Пенев, Ст. Промишлена вентилация и обезпрашаване, София.)*, used as a component of local exhaust ventilation.

EXPOSITION

Build a geometric model

A geometrical model is used, build in 3D modeling software SolidWorks, shown on figure. 1. With position 1 is depicted the inlet air duct, supplying the turbulent jet to a suction nozzle, marked with number 2. The suction air duct is given on position 3. The presence of a certain angle between the sections is indicated by „α" , and the distance between suction nozzle and inlet air duct with L.

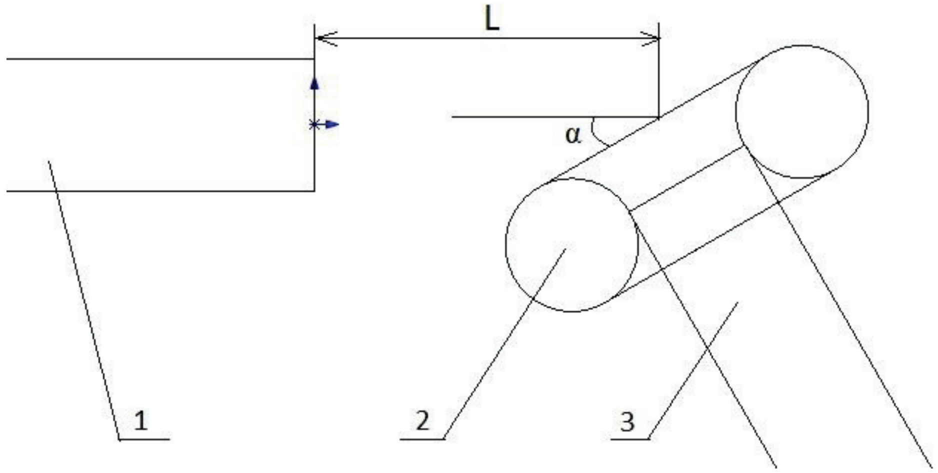

Figure 1. Geometric model for the study of the turbulent jet interaction with the suction opening in the presence of an angle between the sections

The equations that mathematically model the air flow are known and used in Ansys Fluent and according to *(Antonov I. (2016). Prilozhna mehanika na fluidite, Sofiya)*, have the following form:

$$\rho\frac{\partial u}{\partial t}+\rho u\frac{\partial u}{\partial x}+\rho v\frac{\partial u}{\partial y}+\rho w\frac{\partial u}{\partial z}=-\frac{\partial p}{\partial x}+\mu\left(\frac{\partial^2 u}{\partial x^2}+\frac{\partial^2 u}{\partial y^2}+\frac{\partial^2 u}{\partial z^2}\right)+\frac{1}{3}\mu\frac{\partial}{\partial x}\left(\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}+\frac{\partial w}{\partial z}\right)+ \mu_t\left(\frac{\partial^2 u}{\partial x^2}+\frac{\partial^2 u}{\partial y^2}+\frac{\partial^2 u}{\partial z^2}\right)+\frac{1}{3}\mu_t\frac{\partial}{\partial x}\left(\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}+\frac{\partial w}{\partial z}\right) \tag{1}$$

$$\rho\frac{\partial v}{\partial t}+\rho u\frac{\partial v}{\partial x}+\rho v\frac{\partial v}{\partial y}+\rho w\frac{\partial v}{\partial z}=-\frac{\partial p}{\partial y}+\mu\left(\frac{\partial^2 v}{\partial x^2}+\frac{\partial^2 v}{\partial y^2}+\frac{\partial^2 v}{\partial z^2}\right)+\frac{1}{3}\mu\frac{\partial}{\partial y}\left(\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}+\frac{\partial w}{\partial z}\right)+ \mu_t\left(\frac{\partial^2 u}{\partial x^2}+\frac{\partial^2 u}{\partial y^2}+\frac{\partial^2 u}{\partial z^2}\right)+\frac{1}{3}\mu_t\frac{\partial}{\partial y}\left(\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}+\frac{\partial w}{\partial z}\right) \tag{2}$$

$$\rho\frac{\partial w}{\partial t}+\rho u\frac{\partial w}{\partial x}+\rho v\frac{\partial w}{\partial y}+\rho w\frac{\partial w}{\partial z}=-\frac{\partial p}{\partial z}+\mu\left(\frac{\partial^2 w}{\partial x^2}+\frac{\partial^2 w}{\partial y^2}+\frac{\partial^2 w}{\partial z^2}\right)+\frac{1}{3}\mu\frac{\partial}{\partial z}\left(\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}+\frac{\partial w}{\partial z}\right)+ \mu_t\left(\frac{\partial^2 u}{\partial x^2}+\frac{\partial^2 u}{\partial y^2}+\frac{\partial^2 u}{\partial z^2}\right)+\frac{1}{3}\mu_t\frac{\partial}{\partial z}\left(\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}+\frac{\partial w}{\partial z}\right) \tag{3}$$

The continuity equation is added:

$$\frac{\partial(\rho u)}{\partial x}+\frac{\partial(\rho v)}{\partial y}+\frac{\partial(\rho w)}{\partial z}=0 \tag{4}$$

The system of equations (1÷4) is limited to the following characteristic equation:

$$\frac{\partial}{\partial t}(\rho\Phi) + div(\rho V\Phi) = div(\Gamma grad\Phi) + S \quad , \qquad (5)$$

where Φ is dependent variable; Γ – diffusion coefficient of Φ; S – source part of of the respective Φ. Their values are given at table 1.

Table 1 Values of Γ and S for each Φ

Φ	Γ	S
1	0	0
u	$\mu + \mu_t$	$\Gamma\left(\frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2} + \frac{\partial^2 u}{\partial z^2}\right) - \frac{\partial p}{\partial x} + \frac{1}{3}\frac{\partial}{\partial x}\Gamma\left(\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z}\right)$
v	$\mu + \mu_t$	$\Gamma\left(\frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2} + \frac{\partial^2 u}{\partial z^2}\right) - \frac{\partial p}{\partial y} + \frac{1}{3}\frac{\partial}{\partial y}\Gamma\left(\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z}\right)$
w	$\mu + \mu_t$	$\Gamma\left(\frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2} + \frac{\partial^2 u}{\partial z^2}\right) - \frac{\partial p}{\partial z} + \frac{1}{3}\frac{\partial}{\partial z}\Gamma\left(\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z}\right)$

To close the equation system, to model the turbulent viscosity „μ_t" is used standard k-ε turbulence model, which is based on transport equations for turbulent kinetic energy „k" and dissipation rate „ε", according equations (6) and (7):

$$\rho\frac{Dk}{Dt} = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right] + G_k - \rho\varepsilon \qquad (6)$$

$$\rho\frac{D\varepsilon}{Dt} = \frac{\partial}{\partial x_j}\left(\frac{\mu_t}{\sigma_\varepsilon}\frac{\partial\varepsilon}{\partial x_j}\right) + (C_{1\varepsilon}G_k - C_{2\varepsilon}\varepsilon)\frac{\varepsilon}{k} \quad , \qquad (7)$$

where:

Gk – generation of turbulence kinetic energy due to the mean velocity gradients,

$\mu_t = C_\mu \rho k^2/\varepsilon$ – turbulent viscosity. The standard values of the constants in the equations are:

$C_\mu = 0.09$ $C_{\varepsilon 1} = 1.44$ $C_{\varepsilon 2} = 1.92$ $\sigma_k = 1$ $\sigma_\varepsilon = 1.3$

Simulation of processes is made in Fluent module of software product Ansys [Ansys Fluent 12.0 User's guide, 2009]. The solution is reached after approximately 380 iteractions, according to the predefined criteria.

Results of the numerical solution

The present paper examines three variants of interaction of turbulent jet with suction opening with the presence of an angle between the sections "α" from 30° to 90°, at a constant distance between the sections „L" of 250 mm. The output data for the three options considered is shown in tabular form. By keeping the dependence of the suced flow rate from the nozzle to inlet flow ratio Q_c/Q_H = 1,7.

The boundary conditions for waging numerical simulations are given in table 2.

Table 2: output data for the three options of interaction of turbulent jet with suction opening

Variant 1 L=250mm, α=30°			Variant 2 L=250mm, α=60°		
Name	**Value**	**Dimensionality**	**Name**	**Value**	**Dimensionality**
Inlet duct			Inlet duct		
Diameter D_H	100	mm	Diameter D_H	100	mm
Lenght	300	mm	Lenght	300	mm
Suction duct			Suction duct		

Diameter Dc	100	mm	Diameter Dc	100	mm
Lenght	1900	mm	Lenght	1900	mm
Distance between sections, L	250	mm	Distance between sections, L	250	mm
Jets parameters			Jets parameters		
Inlet flow rate, Q_H	0,0852	kg/s	Inlet flow rate, Q_H	0,0846	kg/s
Suction flow rate, Qc	0,14988	kg/s	Suction flow rate, Qc	0,15012	kg/s
Flow rate relationship: Qc/Q_H	1,76		Flow rate relationship: Qc/Q_H	1,77	
Eccentricity	0	mm	Eccentricity	0	mm
Angle between sections, α	30	grad	Angle between sections, α	60	grad

Variant 3 L=250mm, α=90°		
Name	**Value**	**Dimensionality**
Inlet duct		
Diameter D_H	100	mm
Lenght	300	mm
Suction duct		
Diameter Dc	100	mm
Lenght	1900	mm
Distance between sections, L	250	mm
Jets parameters		
Inlet flow rate, Q_H	0,0792	kg/s
Suction flow rate, Qc	0,14064	kg/s
Flow rate relationship: Qc/Q_H	1,78	
Eccentricity	50	mm
Angle between sections, α	90	grad

Velocity fields, obtained from the described variants are shown on figures 2-4, with sections on the leakage axis. The current lines clearly demonstrate the turbulent character of the flow and reducing the completeness of capture with increasing the angle between main (contaminated) jet and the suction nozzle.

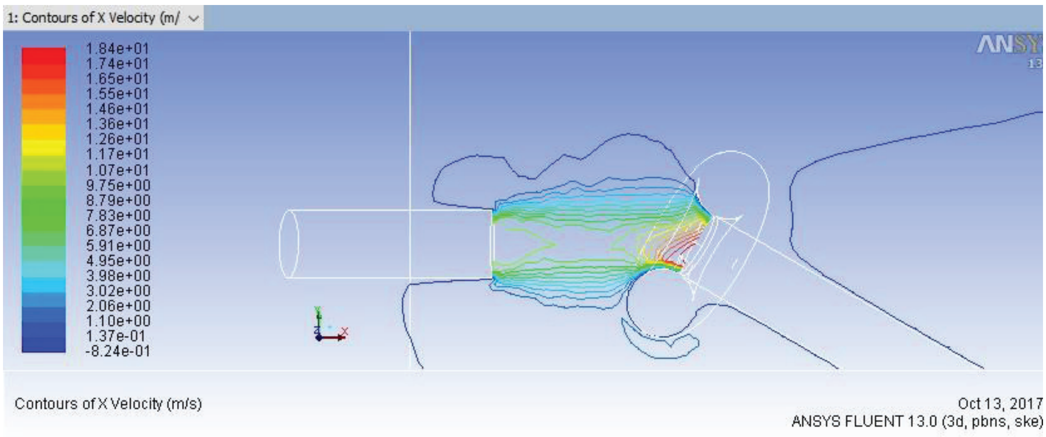

Figure 2. Current lines of the jet at a distance L=250 mm and α=30°

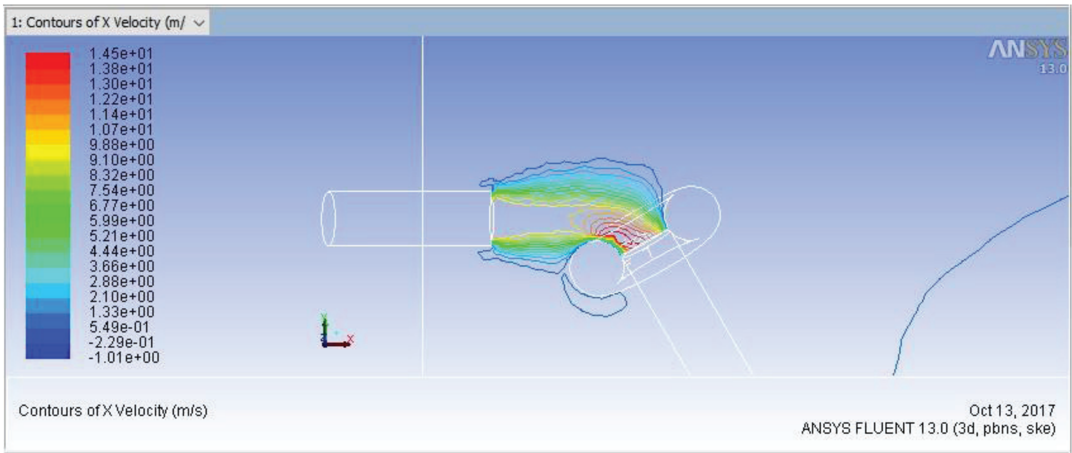

Figure 3. Current lines of the jet at a distance L=250 mm and α=60°

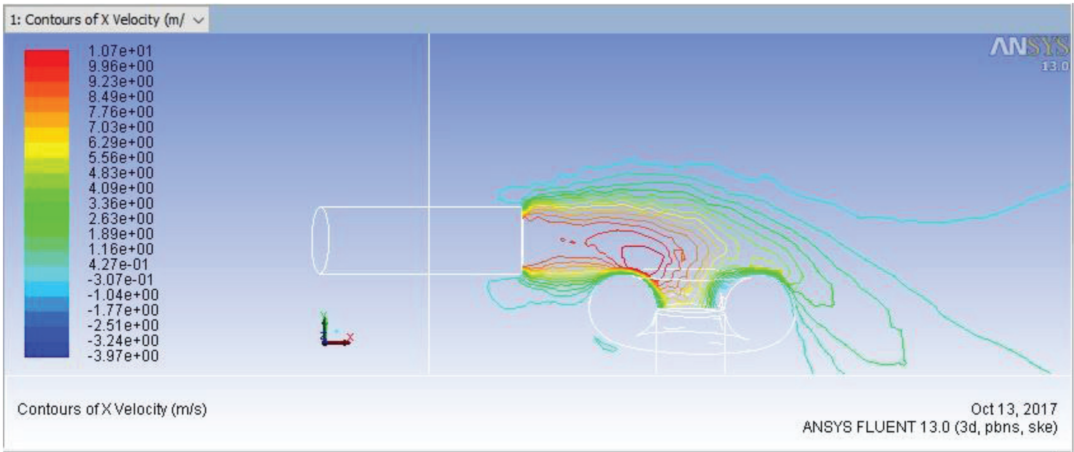

Figure 4. Current lines of the jet at a distance L=250 mm and α=90°

Speed profiles in characteristic sections, longitudinally to the flow, are considered and shown on figures 5-7.

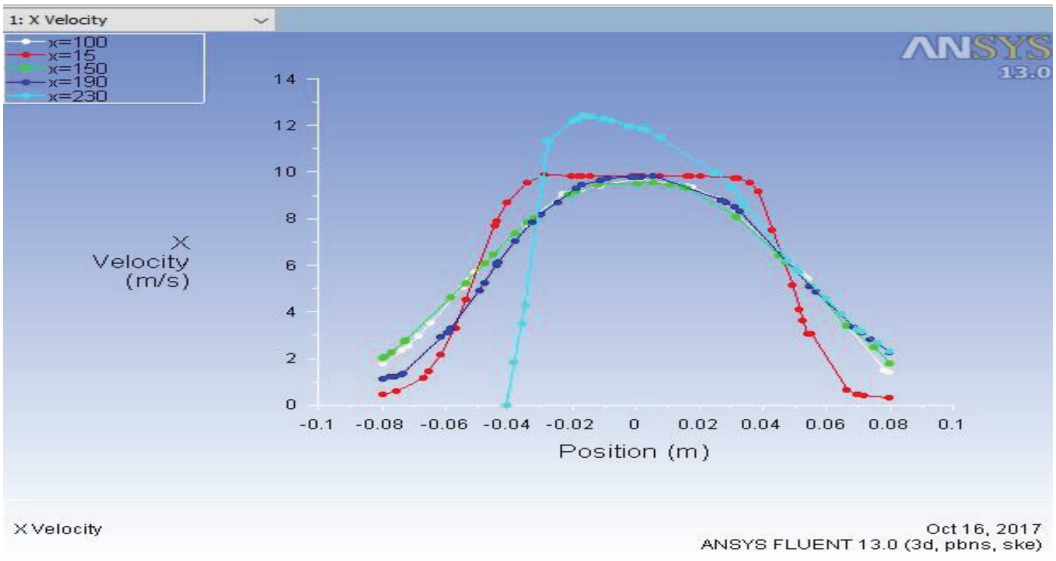

Figure 5. Velocity profiles of the jet at a distance L = 250 mm and α=30°

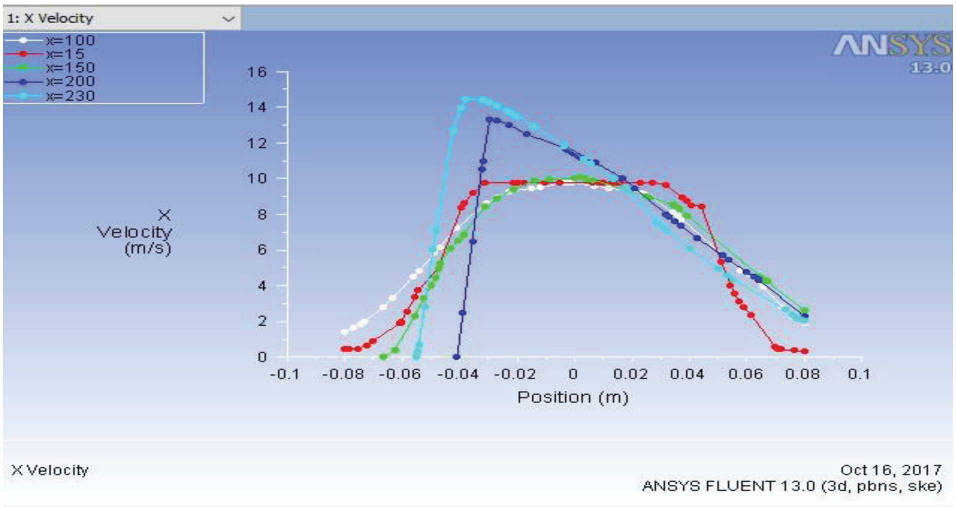

Figure 6. Velocity profiles of the jet at a distance L = 250 mm and α=60°

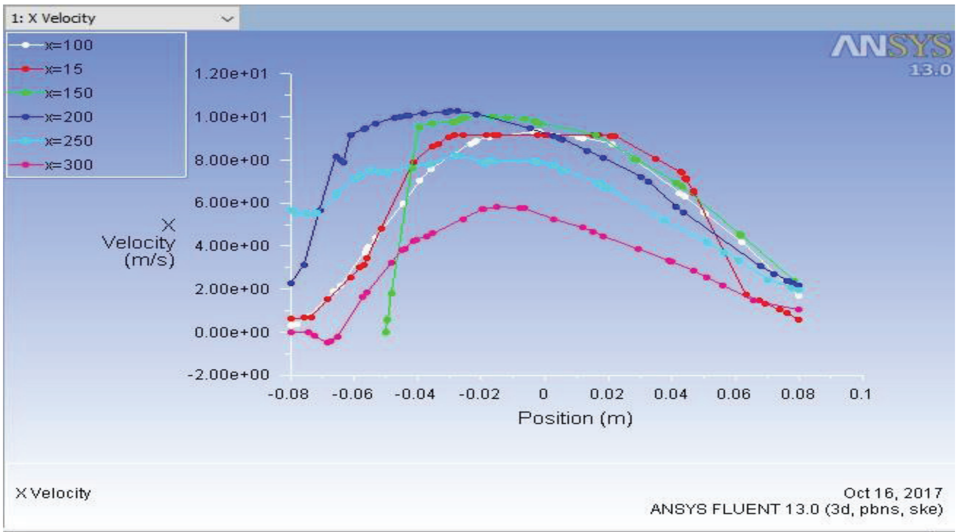

Figure 7. Velocity profiles of the jet at a distance L = 250 mm and α=90°

CONCLUSION

The numerical simulations of interaction of a turbulent jet with suction opening, using CFD modelling gives results, which easily can guide the designers what is going to be the relation between suction flow rate and the main (contaminated) flow rate Qc/Qн, at different angles between sections to achieve full evacuation from the suction air duct. This condition is also a guarantee to prevent harmful particles to remain in the working area and thus increase the concentration above the allowable for health and safety working conditions.

REFERENCES

Ansys Fluent 12.0 User's guide, 2009

Antonov I. (2016). Prilozhna mehanika na fluidite, Sofiya (***Оригинално заглавие:*** *Антонов, И., 2016. Приложна механика на флуидите, София.)*

Olander L., Industrial ventilation design guidebook, Chapter 10 Local Ventilation

Penev S. (2001). Promishlena ventilatsiya I obezprashavane, Sofiya (***Оригинално заглавие:*** *Пенев, Ст. Промишлена вентилация и обезпрашаване, София.)*