MATEC Web of Conferences **68**, 08001 (2016) DOI: 10.1051/matecconf/20166808001
ICIEA 2016

Research on Flow Field Characteristics Affected by Obstacles of Pull-Push Ventilation System of Plating Tanks through Numerical Simulation

Hu Hong[1,2] and Yi Can-nan[2]

[1] *Human Factors Institute, University of South China, Hengyang, China*
[2] *Ergonomics and safety management Institute, HuNan Institute of Technology, Hengyang, China*

Abstract. Pull-push ventilation system, with high capture efficiency, is universally used to capture contaminants from containment generation where can't be enclosed and where the working area is large. When there are obstacles between the pull hood and the push hood, the air jet will disperse, leaving contaminants out of control, causing atmosphere polluted. This paper aims to discuss flow filed of pull-push ventilation system, located in some tank of some plating workshop, with obstacles between pull hood and push hood. Mathematical model, involving physical model, assumptions, governing equations, boundary conditions and grid separation is established. By CFD code FLUENT, flow field, containing velocity field and concentration field, is achieved. To verify the simulation models, velocity field for simulation results is compared with the theory results, while concentration field for simulation results with experimental results. The comparison results show that the results matches well and that mathematical model proposed in this paper is reasonable, helpful and realistic significant in designing this kind exhaust ventilation systems where obstacles presence.

1 Introduction

Push-pull ventilation system, involving a jet of air that is blown from one side of the tank and collected by an exhaust hood on the opposite side, is such empowered local exhaust hoods with significant advantage over local ventilation systems in capturing contaminants and resisting disturbing airflow that they are widely used to control contaminants derived from containment generation where can't be enclosed owing to the necessary to produce the products, nor could other hoods be chosen due to the large working areas. When controlling pollutant, push-pull ventilation system can save up to 50% of flow rate than lateral exhaust [1], and it is prior to any other hoods when the hood is far away from pollutant generation (eg, more than 2m). Nowadays, push-pull ventilation system is widely chosen to capture harmful gas and vapor [2] and fumes [3]. Since it controls the pollutants through the associative action of pull hood and push hood, we should determine the best effective and matched velocities of them. It is suggested use numerical methods to analyze flow field instead of experimental work which can't easily be extended to different operating conditions [4]. Michael Robinson present a semi - analytical approach which exploits the self - similar nature of much of the fluid flow [5], R. Rota discussed the design guidelines for push-pull ventilation systems through computational fluid dynamics modeling [6], Hunt [7], HE Su-yan [8] and CHEN Jiang-ping [9] deal with the mathematical model, flow-field test, and tuburlence character. Existing simulation methods only focus on the flow character of the hoods or big obstacles between hoods (desk, computer, chair redistribute [10], ceiling mounted obstacles [11] or woker [12]), nor do they consider that there are some small obstacles existed between pull hood and push hood located in plating tanks. When there are obstacles, the matched velocities of the two hoods become crucial for us to determine: if they are slower than the demand, the containments won't be captured entirely; while they are faster, the air jet will be dispersed to outer space when it shot the obstacles.

This paper aims to analyze the flow field of push-pull ventilation system over plating tanks with obstacles between them: setting the pollutant gas emitted on plated pieces, with volatile liquid adhesives from the tanks, of sulfate electroplating tinning product line of some factory as the physical model, combining with some theories, such as hydrodynamic, aerodynamics and so on, mathematical model is built based on some assumptions; by CFD commercial code FLUENT, flow field, including velocity field and concentration field, of the push-pull ventilation system with various matched velocities is simulated, helping design this kind of ventilation system.

2 Mathematical models

2.1 Physical model

Harmful gases generated in the plating process due to the high concentrations of liquid adhered over the plated pieces, moving among the tanks with cranes, will pose great potential hazard to workers and environment. The main gas in plating line of some factory is NO_2 given off by pickling tanks, with high concentrations of nitracidium, a mixture of nitrate solution and water with the proportion of 1:3. We find that the acid mist produced on moving plated pieces is 800～3 000 g/(m^2·h), so the maximum value3 000 g/(m^2·h), seeing as a constant value and not varying with time, is chosen as the target contaminant we should capture. Related parameters are set as follows: mass flow (mass source term)is 0.000 084 kg/(m^3·s), while momentum source term 0.000 055 8 N/m^3(momentum equals to velocity times mass flow), summer, room temperature T=310K, tank liquid temperature T=313 K, simulating the near-source zone only, physical model, as shown in Fig. 1, is enlarged up to 3.5×3×3 m^3 to neglect the outer space's influence.

2.2 Assumptions

1)Steady flow field.

2)Neglecting the density variations of the contaminants and air when velocity is quite slow, namely, seeing them as incompressible fluid.

3)Exception of gravity effects.

4)Assuming that the selected control volume is large enough when adopt control volume method to calculate.

5)Treating the hoods as wall and considering it is radiationless and thermal isolated.

6)Pollutant diffuses evenly.

7)The plated pieces is simplified as cuboids shown in Fig. 2.

8)Neglecting the influence of the plating crane to the flow field.

9)When plating, the crane moves slowly, with the velocity of 0.2 m/s. Push-pull hood located in carne moves with plated pieces simultaneously, remaining relatively static.

10)Setting the obstacles as wall at ordinary temperature , emitting contaminants with temperature of 313K and velocity of 0.000 055 8 N/m^3.

11)Ignoring the disturbing air currents.

2.3 Governing equations

There are many equations over simulating contaminants emissions [13]: standard $k-\varepsilon$ turbulence model realizable $k-\varepsilon$ turbulence model, and large-eddy simulation(LES). HE Su-yan found that the standard $k-\varepsilon$ turbulence model works well when simulating contaminants. So we choose standard $k-\varepsilon$ turbulence mode. General governing equation is [14], [15]:

$$div(\rho \dot{u}\phi) = div(\Gamma grad\phi) + S$$

where ϕ is general variable, Γ diffusion coefficient, S source term. Every symbol and its corresponding form is as Table 1 shows.

2.4 Boundary conditions

Inlet boundary: velocity-inlet

Outlet boundary (face of the open space where push hood exists): pressure-outlet

Outlet boundary (push hood): outlet-vent

Outlet boundary(faces of open space): pressure-inlet

Interior surface boundary: auxiliary surface, established by separating grid, is defined as interior surface where flow can pass through freely

Source term (plated pieces): According to the results of in-situ measurements, mass flux of NO_2 is 8.4×10^{-5} kg/(m^3·s), with the momentum flux of 5.58×10^{-5} N/m^3.

Table 1. Detail forms for the symbol of general governing equations

Equations	ϕ	Γ	S
Mass conservation equations	1	0	0
Momentum equations	u_i	$\mu_{eff} = \mu + \mu_t$	$-\frac{\partial P}{\partial x_i} + S_i$
Species mass-conservation equations	c_s	$D_s \rho$	S_s
Turbulent kinetic energy	k	$\mu + \frac{\mu_t}{\delta_k}$	$G_k + \rho\varepsilon$
Turbulent dissipation rate	ε	$\mu + \frac{\mu_t}{\delta_\varepsilon}$	$\frac{\varepsilon}{k}(C_{1\varepsilon}G_k - C_{2\varepsilon}\rho\varepsilon)$
Where : $\mu_t = \rho C_\mu \frac{k^2}{\varepsilon}$, $G_b = \beta g_i \frac{\mu_t}{Pr_t}\frac{\partial T}{\partial x_i}$, $G_k = \mu_t\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right)\frac{\partial u_i}{\partial x_j}$, $Y_M = 2\rho\varepsilon M_t^2$,			
Constant numbers [16], [17] : Pr_t=0.85, $C_{1\varepsilon}$=1.44, $C_{2\varepsilon}$=1.92, C_μ=0.09, σ_k=1.0, σ_ε=1.3			

2.5 Grid Separation

The computational domain is 3.5×3×4.5m^3, hood 0.4×0.6 m^2, columned plated pieces with 0.05m in diameter and 0.4m in height. To simplify the calculation, little columned plated pieces is treated as a little cuboids, with 0.055×1.8×0.4 m^3, defined as wall. The space expect hoods and cuboids is divided into 2 fluid zones (Fig. 2): fluid zone 1 is a very thin shield of flow, circling the cuboids, which is assumed to be contaminants generation, with mesh interval of 0.025m; fluid zone 2 is the residue space. The mesh interval of the hoods is 0.1m. The face is separated by pave triangle grid and the volume Tgrid. Partial grid near hoods and plated pieces are separated with shorter intervals to enhance the precision and convergence speed of calculation, and the whole computational domain is separated into 300,000 computation cells as Fig. 3 shows.

 DOI: 10.1051/matecconf/20166808001

3 Simulation Results and Flow Field Profile

FLUENT code is used to analyze push-pull ventilation system's flow field. The matched velocities of push unit and pull unit calculated by flow ratio method are: v_1=2 m/s, v_3=10.42 m/s; v_1=2.5 m/s, v_3=11.67 m/s; v_1=3 m/s, v_3=12.74 m/s; v_1=3.5 m/s, v_3=14.0 m/s, respectively. Velocity contours and mass fraction contours are as shown from fig. 4 to fig.7.

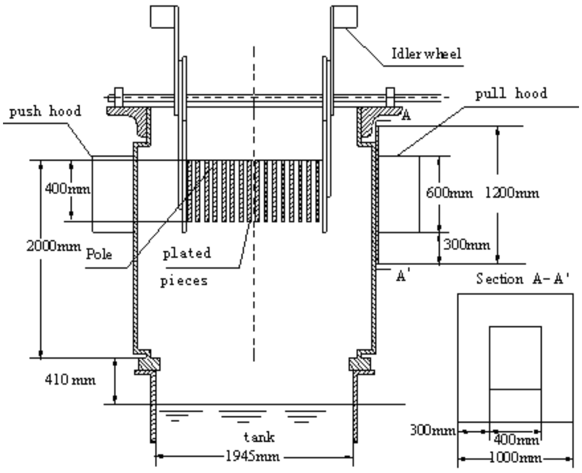


Figure 1. Pull-push ventilation system fixed on crane

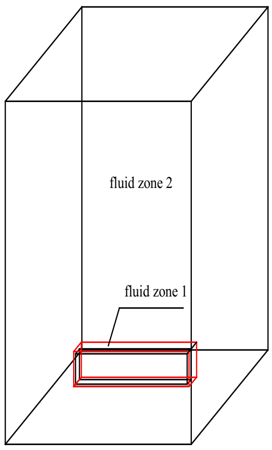


Figure 2. Fluid zones

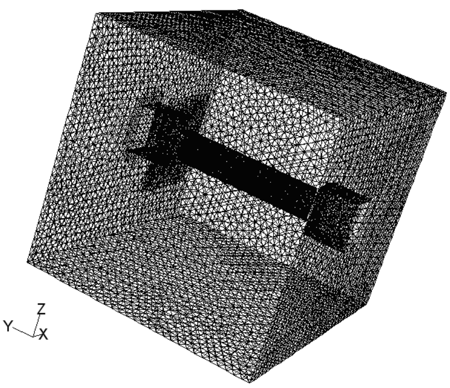
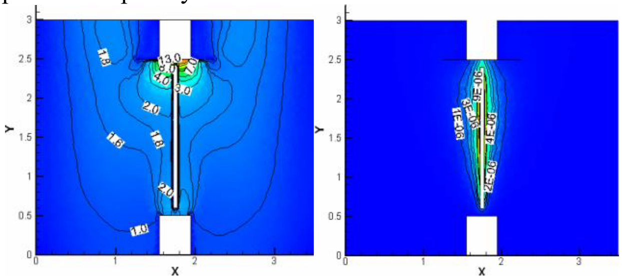


Figure 3. Schematics of grid separation

1)Velocity contours

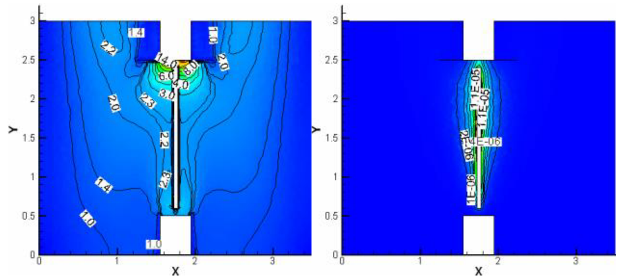
(1)The air jet disperses due to the obstacles, the greater the velocity is, the more obvious the dispersion is.

(2)The best matched velocities are v_1=2.5 m/s and v_3=11.67 m/s, forming a gas curtain covering the plated pieces completely with the smallest air flow.



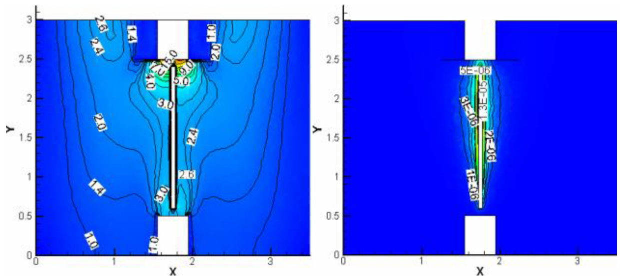
(a) Velocity contours (b) NO_2 mass fraction contours

Figure 4. Velocity contours and NO_2 mass fraction contours at section *z*=1.5 m (v_1=2 m/s)



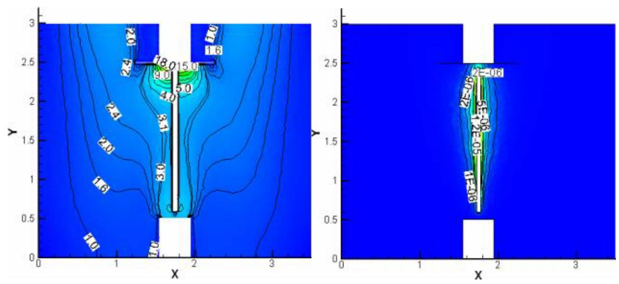
(a) Velocity contours (b) NO_2 mass fraction contours

Figure 5. Velocity contours and NO_2 mass fraction contours at section *z*=1.5 m(v_1=2.5 m/s)



(a) Velocity contours (b) NO_2 mass fraction contours

Figure 6. Velocity contours and NO_2 mass fraction contours at section *z*=1.5 m(v_1=3 m/s)



(a) Velocity contours (b) NO_2 mass fraction contours

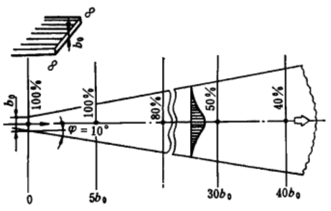
Figure 7. Velocity contours and NO2 mass fraction contours at section z=1.5 m(v1=3.5 m/s)

2)NO_2 mass fraction contours

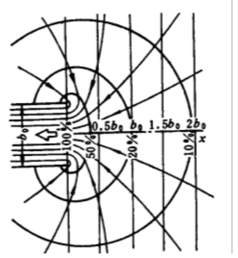
From Fig. 4 to Fig. 7, we can find that the greater the velocity is, the closer the contours presses to the plated pieces, and the smaller the contaminants emitting radius is. Obviously, the larger the air flow rate is, the higher the contaminants capture efficiency is. But large air flow means higher costs, including equipments outlays, operation costs and maintenance costs. What we pursued is the smallest air flow rate with contaminants captured entirely. From this point of view, when contaminant emits at 0.5 m/s, the velocity of push hood should be greater than or equal to 2.5 m/s.

4 Comparative Analysis for Results

4.1 Velocity field comparison



(a) 2D velocity profile of blowing air flow



(b) 2D velocity profile of drawing air flow

Figure 8. Velocity profile correlation for pulling and pushing air flow

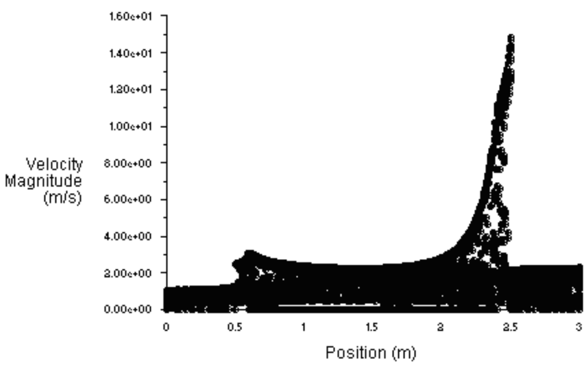


Figure 9. Velocity profile along the x coordinate axis at section z=1.5m

Fig. 8 [18] is the universally accepted velocity profile schematics for push jet and pull air flow which attenuate with the increase of distance. As we can see from the figure that velocity of push jet remains unchanged within $5b_0$ away from push hood; while the velocity of pull air flow attenuates to be 20% of face velocity at b_0, that is, the velocity at pull hood would be 12m/s if the velocity at b_0 was 2.4m/s. Fig. 9 is the schematics for velocity profile along the x coordinate axis at section z=1.5m, the push hood lies in the location of x=0.4, while the pull hood x=2.6. It is easily to be found that the push jet velocity attenuates slowly, the velocity at x=1.6, that is $3b_0$ away from push hood, keeps in 2.5m/s, which matches closely with the results from Fig. 8; while the induced air velocity of pull hood attenuates rapidly, the velocity at x=2.2m, that is b_0 away from drawing cover, attenuates to be 2.5m/s, which basically matches with the results from Fig. 8. Therefore, the simulation results appear to be reasonable.

4.2 Concentration field comparison

To verify the simulation models, combining with BW–GAXT NO_2 tester, NO_2 concentration at face z=1.5 m with location A(1.6,0.8), B(1.6,1.6), C(1.6,2.4), D(2,0.8), E(2,1.6), F(2,2.4), as shown in Fig. 10, and velocities of I(v_1=2 m/s, v_3=10.42 m/s), II(v_1=2.5 m/s, v_3=11.67 m/s), III(v_1=3 m/s,v_3=12.74 m/s), IV(v_1=3.5 m/s, v_3=14.0 m/s), are tested (Table 2). From this table, we can find that simulation values of NO_2 concentration nearly keeps with experimental data, with errors less than 9%.

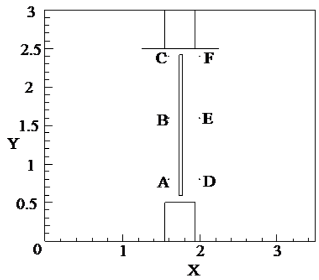


Figure 10. Diagrammatic sketch of detecting position of NO_2

Table 2. simulation values and experimental data of NO_2 concentration at z=1.5m×10^{-6}

	I		II		III		IV	
	SV	ED	SV	ED	SV	ED	SV	ED
A	1	1.3	1	1.2	1	1	1	1
B	9	10	4	5	3.3	3	2	1.8
C	2	2.2	2	2.1	2	2.1	1.6	1.7
D	0	1	0	1	0	0	0	0
E	2	3	1	2	0	0	0	0
F	0	1	0	1	0	0	0	0

Where : SV :simulation values , ED :experimental data

5 Conclusions

(1) Taking the volatile contaminants given off in plating process of some enterprise as subject, combining with some assumptions, mathematical model is established. By CFD code FLUENT, flow field, under v_1=2 m/s, v_3=10.42 m/s; v_1=2.5 m/s, v_3=11.67 m/s; v_1=3 m/s, v_3=12.74 m/s; v_1=3.5 m/s, v_3=14.0 m/s, of pull-push ventilation system, over plating tank, with obstacles between them is obtained. When obstacles presenting, air jet from push hood will disperse. Our research find that in this circumstance the best matched velocities are v_1=2.5 m/s and v_3=11.67 m/s.

(2) Computer simulation is useful to simulate pull-push ventilation system's performance under various conditions, and benefit to determine the best matched velocities of push air flow and pull air flow to avoid equipment costs due to the change of the size, leaving less experimental cost and shortened experimental period.

(3) Results show that simulated velocity field matches accepted theory well, while simulated concentration field keeps with experimental data nearly. So mathematical model presented in this paper is helpful for design.

Acknowledgements

The research presented in this paper is supported by Hengyang scie-tech Bureau (2014KG65), leading academic discipline project of Hunan Institute of technology (HXK13002).

References

1. Malin B S. Practical pointers on industrial exhaust system. Heat & Vent, **42**: 75–82(1945).
2. Enrique Gonza¡ä lez, Francisco Marzal, Agustı¡än Min˜ ana,etc. Influence of Exhaust Hood Geometry on the Capture Efficiency of Lateral Exhaust and Push–Pull Ventilation Systems in Surface Treatment Tanks. Environmental Progress, **27**(3):405-411, (2008).
3. Yi Zheng, Magesh Thiruvengadam, Hai Lan. Design of push–pull system to control diesel particular matter.inside a dead-end entry. Int J Coal Sci Technol, **2**(3):237–244 (2015).
4. Guo Peishan, Fu Haiming, Zhao Youjun, et al. Development and present status of push-pull ventilation technology. Journal of HV & AC, **38**(4): 57–61 (2008).
5. Michael Robinson, Derek B. Ingham. Design and operating parameters for push - pull ventilation of open surface tanks. International Journal of Energy Research, **27** (8):757-770(2003).
6. R. Rota, G. Nano, L. Canossa. Design Guidelines for Push-Pull Ventilation Systems through Computational Fluid Dynamics Modeling. AIHAJ: A Journal for the Science of Occupational & Environmental Health&Safety. **62** (2):141-148.(2001)
7. Hunt G R, Ingham D B. Long range exhaustion--a mathematical model for the axisymmetric air flow of a local exhaust ventilation hood assisted by a turbulent radial jet. The Annals of Occupational Hygiene, **40**(2): 171–196(1996).
8. He Suyan, Wang Yingmin, Li Yunfei. Flow-field test and simulation of a three-dimensional Aaberg exhaust hood. Journal of Shanghai Jiaotong University, **36**(10): 1492–1495(2002).
9. Chen Jiangping, Feng Xin, Mu Jingyang. Numerical simulation of turbulence character the 2-layer non-isothermal blow-draw air-curtain. Journal of Refrigeration, **4**: 16–20 (2001).
10. Nielsen PV. Computational fluid dynamics and room air movement. Indoor Air, **14**:134–143 (2004).
11. K Svidt. Numerical prediction of airflow in a room with ceiling-mounted obstacles.Denmark, Aalborg Universitets center, (1992).
12. Welling I, Andersson M, Rosen G, Ra¨isa¨nen J, Mielo T, Marttinen K, Niemela¨ R Contaminant dispersion in the vicinity of a worker in a uniform velocity field. Ann Occup Hygiene, **44**(3):219–225 (2000).
13. Xian Xiangli, Chun Holiu, Dennis Y.C.Leung, K.M.Lam. Recent progress in CFD modeling of wind field and pollutant transport in street canyons. Atmospheric Environment, **40**:5640-5658(2006).
14. J.D. Anderson, McGrawHill.Computational Fluid Dynamics: The Basics with Application. Tinghua University Press (1995).
15. Kambiz Nazridoust, Goodzarz Ahmadi. Airflow and pollutant transport in street canyons. Journal of wind Engineering and Industrial Aerodynamics, **94**:491-522 (2006).
16. Tao Wenquan.Numerical Heat Transfer (the 2nd edition).Xi'an: Xi'an Jiaotong University Press (2002).
17. Wang Fujun. Computational Fluid Dynamics. Beijing: Tsinghua University Press (2004).
18. Sun Yijian. Industry Ventilation (the 3rd edition). Beijing: China Building Industry Press (2000).