Applied Thermal Engineering 110 (2017) 573–583

Contents lists available at ScienceDirect

Applied Thermal Engineering

journal homepage: www.elsevier.com/locate/apthermeng

Research Paper

# Design method and modeling verification for the uniform air flow distribution in the duct ventilation

Wei-Biao Ye

*Department of Process Equipment and Control Engineering, School of Mechanical Engineering, Xiangtan University, Xiangtan 411105, PR China*

## HIGHLIGHTS

- The design method for the uniform air flow distribution is presented.
- The modeling verification is conducted by introducing the practical case.
- The unbalance rate of air flow distribution is less than 10%.
- The vertical baffles can improve air flow distribution inside the fixtures further.

ARTICLE INFO

*Article history:*
Received 23 May 2016
Revised 25 August 2016
Accepted 27 August 2016
Available online 30 August 2016

*Keywords:*
Uniform air flow distribution
Duct ventilation
Design method
Numerical verification

ABSTRACT

Uniform air flow distribution is an important and fundamental issue to miscellaneous equipments. The detailed design method and design principle are presented in this work. The practical application for uniform air flow distribution is conducted according to the employed design method. Finite volume method (FVM) is used to verify the designed validation. The numerical treatment of convection terms in governing equations is based on the quadratic upwind interpolation of convective kinematics (QUICK) scheme. The well-know semi-implicit pressure-linked equation (SIMPLE) algorithm is used to treat the coupling of pressure and velocity fields. It is found that the design method is validated for the uniform air flow distribution. The unbalance rate of air flow distribution is less than 10%. Moreover, the uniform of air flow distribution can be further improved by adding vertical baffles. The developed design method is of great significance to obtain uniform air flow distribution.



## 1. Introduction

A group of European scientists has reviewed scientific papers on ventilation and the effects on health, comfort, and productivity in offices, schools and homes. The group found that the degree of ventilation was strongly linked to perceive air quality and healthy [1]. Recently, the ventilation studies were conducted including buoyancy-driven natural ventilation investigated by Hussain and Oosthuizen [2], they explored numerically the solar-assisted buoyancy-driven natural ventilation for thermal comfort conditions in the building. It was concluded that the proposed method can reliable thermal comfort predictions. Furthermore, Yang et al. [3] investigated the natural ventilation effectiveness of office space in common public building used computational fluid dynamics (CFD) simulations and field measurements. It was shown that the CFD simulation procedure can serve as a useful tool to facilitate architects for improving the natural ventilation design of buildings. Additionally, Pasut and Carli [4] evaluated the various CFD modeling strategies in naturally ventilated double skin façade. The results shown that the additional effort required to make a three-dimensional model was not justified by a significant improvement of the results. In addition to this, Beiza et al. [5] presented an algebraic thermal zonal model of the ventilation of underground transformer substations by numerically solved used CFD techniques. They inferred that the main parameters affecting the ventilation of the substations were the pass area, the surface area of the ventilation grilles and the perimeter of the protruding ventilation vents. Beside natural ventilation as mentioned in Refs. [2–5], Ai and Mak [6] for the first time developed a short-term mechanical ventilation strategy, which can determine appropriate design parameters, including ventilation period, ventilation frequency, and start concentration of ventilation. They divided a whole sleeping period of 8 h into many repeated single *V*-shape ventilation periods. As a result, a high efficient ventilation strategy was a short single ventilation period and a high ventilation frequency. Regarding the applications for passive ventilation, Connor et al. [7] conceptualised a novel design of a desiccant rotary wheel

*E-mail addresses:* weibiaoye@xtu.edu.cn, weibiaoye@foxmail.com

http://dx.doi.org/10.1016/j.applthermaleng.2016.08.177
1359-4311/

**Nomenclature**

| | |
|---|---|
| $D$ | hydraulic diameter, m |
| $e$ | roughness, m |
| $f$ | friction resistance coefficient |
| $g$ | gravity acceleration, m/s$^2$ |
| $h$ | duct height, m |
| $L$ | length, m |
| $n$ | total nozzles number |
| $P_d$ | dynamic pressure drop, Pa |
| $P_l$ | local pressure loss, Pa |
| $P_s$ | static pressure drop, Pa |
| $\dot{Q}_0$ | air flow volume rate through single hole, m$^3$/s |
| $\dot{Q}_f$ | air flow volume rate through the front cross section of hole, m$^3$/s |
| $\dot{Q}_{total}$ | total air flow volume rate, m$^3$/s |
| $Re$ | Reynolds number |
| $S_0$ | hole's area, m$^2$ |
| $S$ | project area of hole's area at air outflow direction, m$^2$ |
| $v_0$ | average velocity of side hole, m/s |
| $u_{i,j}$ | velocity component, m/s |
| $v_d$ | velocity driven by dynamic pressure, m/s |
| $v_r$ | real velocity, m/s |
| $v_s$ | velocity driven by static pressure, m/s |
| $w$ | duct width, m |
| $x_{i,j}$ $(x, y, z)$ | dimension in Cartesian coordinate, m |

*Greek symbols*

| | |
|---|---|
| $\mu$ | dynamic viscosity, kg/m s |
| $\theta$ | outflow angle, ° |
| $\rho$ | density, kg/m$^3$ |
| $\psi$ | flow coefficient of holes |
| $\zeta$ | local resistance coefficient |

*Abbreviations*

| | |
|---|---|
| CFD | computational fluid dynamics |
| FVM | finite volume method |
| QUICK | quadratic upwind interpolation of convective kinematics |
| RANS | Reynolds average numerical simulation |
| SIMPLE | semi-implicit pressure-linked equation |

to control the humidity of the incoming air, the design was verified by CFD and prototype testing.

European ventilation strategies in building heating and cooling were summarized by Guillen-Lambea et al. [8], they reviewed and compared the ventilation flow rates in residential buildings in various European countries. It was found that with the current ventilation strategies, the heating and cooling demand values required by Passivhaus can be reached in only a few warm climates. Furthermore, Calay and Wang [9] presented a high performance cooling/heating hybrid ventilation system, the hybrid system provided heating in the winter and cooling in the summer did not for additional heating or cooling devices as required in conventional systems. The results shown that 60% energy saving was achieved by employed the proposed hybrid system. In addition, a residential hybrid ventilation system was experimentally investigated by Turner et al. [10], the novelty was that the system can provide filtered and tempered fresh air, either to residences or small commercial buildings. In either heating/cooling mode, it was found that the hybrid system was capable of operating effectively to provide an indoor/outdoor air temperature difference of 10 K across a range of air flows up to 120 L/s. Meanwhile, Yu et al. [11] established a detailed quasi-three-dimensional mathematical model of heat and mass transfer of tailrace tunnel ventilation based on the analysis of heat and mass transfer during the air flowing through the tunnel. The conclusions showed that the air condition was gradually close to the saturation water temperature with ignored the effect of long-term thermal cumulative of the tunnel rock. The heat and mass transfer between air and water surface was dominant, which had a ratio of 80–90% in the total heat and mass exchange.

It should be noted that uniform air flow distribution is especially required in many engineering fields, such as in plate-type heat exchangers [12], air conditioning systems [13], heat sinks for electronic devices cooling [14,15], solar thermal collectors [16], piping system [17], chemical reactors [18], and nuclear reactors [19], etc. In modern agricultural irrigation systems, uniform flow distribution provides the maximum production under the minimum use of water [20]. Moreover, in biological field, research showed that uniform flow distribution is helpful to the reduction of flow resistances in vascular system [21]. Furthermore, uniform flow distribution is also important for fuel cells that require uniform electrochemical reactions for high energy conversion efficiency investigated by Li [22], Danilov [23], Wang [24], and Li [25]. It is generally known from all the above-mentioned applications that uniform flow distribution is often advantageous in providing better heat transfer, temperature control, and low pressure loss, which is translated into less pumping power, as well as minimization of flow-related vibrations, noise, thermal and flow stresses, and corrosions due to higher reliability and durability of flow-involved facilities and devices.

However, all of above-mentioned studies do not develop the systemic design method regarding how to realize the uniform air flow distribution. In practical applications, it is common that non-uniform duct ventilation will happen due to constant cross section of air duct. These phenomena will result in high air flow rate at the top level of drying equipments, i.e., the closed end position of the air flow duct in the drying equipments. However, there is a not air flow the bottom level at all. Therefore, the established method with constant cross section of the duct is quite unreasonable. For this point, it is necessary to obtain uniformity duct ventilation for every level inside drying box to heat the material uniformly and reach the same heating time. Therefore, the various cross sections for air flow duct are proposed in the current work. At first, this work describes the detailed design principle and how to realize uniform air flow distribution. The detailed design process is then performed by introducing the practical case. Finally, the numerical verification is conducted to ensure the reliability of design method. The detailed design and numerical verification processes are presented in the following sections in detail.

## 2. Design the uniform air flow distribution

### 2.1. Design principle

Air flow will flow out if the holes existed in the duct due to the static pressure difference between inside wall and outside wall of the holes. Therefore, the direction of air velocity of outflow ($v_s$) is vertical to the duct wall, as shown in Fig. 1. However, the velocity ($v_d$) in duct also influence the real velocity ($v_r$) direction of air flows out of the side holes.