View metadata, citation and similar papers at core.ac.uk
brought to you by CORE
provided by Purdue E-Pubs

Purdue University
Purdue e-Pubs

International High Performance Buildings Conference
School of Mechanical Engineering

2016

The Air Distribution Around Nozzles Based On Active Chilled Beam System

Bingjie Wu
Energy Research Institute @ NTU (ERI@N), Interdisciplinary Graduate School, Nanyang Technological University, Singapore, bwu006@e.ntu.edu.sg

Wenjian Cai
School of Electrical and Electronic Engineering, Nanyang Technological University, 50 Nanyang Avenue, Singapore, 639798, ewjcai@ntu.edu.sg

Qinguo Wang
Institute for Intelligent Systems, the University of Johannesburg, South Africa, wangq@uj.ac.za

Can Chen
School of Electrical and Electronic Engineering, Nanyang Technological University, 50 Nanyang Avenue, Singapore, 639798, cchen009@e.ntu.edu.sg

Chen Lin
School of Electrical and Electronic Engineering, Nanyang Technological University, 50 Nanyang Avenue, Singapore, 639798, linchen@ntu.edu.sg

See next page for additional authors

Follow this and additional works at: http://docs.lib.purdue.edu/ihpbc

Wu, Bingjie; Cai, Wenjian; Wang, Qinguo; Chen, Can; Lin, Chen; and Chen, Haoran, "The Air Distribution Around Nozzles Based On Active Chilled Beam System" (2016). *International High Performance Buildings Conference.* Paper 173.
http://docs.lib.purdue.edu/ihpbc/173

This document has been made available through Purdue e-Pubs, a service of the Purdue University Libraries. Please contact epubs@purdue.edu for additional information.
Complete proceedings may be acquired in print and on CD-ROM directly from the Ray W. Herrick Laboratories at https://engineering.purdue.edu/Herrick/Events/orderlit.html

Authors
Bingjie Wu, Wenjian Cai, Qinguo Wang, Can Chen, Chen Lin, and Haoran Chen

This article is available at Purdue e-Pubs: http://docs.lib.purdue.edu/ihpbc/173

The air distribution around nozzles based on active chilled beam system

Bingjie Wu[1,2], Wenjian Cai[2,*], Qinguo Wang[3],Can Chen[2], Chen Lin[2], Haoran Chen[2]

[1]*Energy Research Institute @ NTU (ERI@N), Interdisciplinary Graduate School, Nanyang Technological University, Singapore*
[2]*School of Electrical and Electronic Engineering, Nanyang Technological University, Singapore*
[3]*Institute for Intelligent Systems,the University of Johannesburg,South Africa*
** Corresponding author. W Cai, Email: EWJCAI@ntu.edu.sg*

Abstract

During the past two decades, the utilization of Active Chilled Beam (ACB) systems as promising air-conditioning systems has becoming increasingly prevalent in Europe, North America and Asia. However, the studies on air distribution of ACB systems are still inadequate. The air distribution has a great impact on thermal comfort. The un-uniformity of air distribution will easily lead to turbulent flow which can cause unpleased feeling such as draught in the occupied zone. ACB terminal unit is the source of the air flow entering into the occupied place, which plays a crucial role on the air distribution inside the room. Therefore, it's of great importance to evaluate the air distribution in the vicinity of ACB nozzles.

In order to fulfil the gap, air distribution for a two-way discharge ACB terminal unit is investigated in this study. The air velocities around the nozzles under different conditions are tested in a 7.3m*3.3m*2.5m thermal isolated room and simulated by a three dimensional Computational Fluid Dynamics (CFD). After being verified, the CFD model is utilized to examine the effects of nozzle diameter and inlet pressure. From the results of experiments and simulation, it is found out that the air flow is discharged in an asymmetric way from nozzles, which is ascribed to un-uniformity of pressure distribution inside ACB caused by the layout of the duct. Moreover, the un-uniformity is significant when the nozzle diameter is large and the elevation of the inlet pressure would aggravate this un-uniformity. Therefore, as we design the ACB systems, high attention on the nozzle diameter should be paid to prevent the un-uniformity air flow when the nozzles and the inlet pressure are large. Eventually, a proper strategy to solve this problem is also proposed and validated by CFD simulation.

Key words: Air distribution, Uniformity, Active chilled beam, CFD, Thermal comfort

1 Introduction

The chilled beam system has been proved to be a reliable Heating, Ventilation and Air Conditioning (HVAC) solution ever since it was introduced in 1980s. It can be classified into Active Chilled Beam (ACB) system and passive chilled beam system according to whether there is ventilation of fresh air. Owing to the advantages of providing good energy efficiency, thermal comfort and noise control with less space requirement, ACB system has been increasingly popular in not only Europe, but also North America and Asia [1]. It's wildly utilized in a variety of commercial buildings, schools, laboratories and hospitals[2].

ACB is the terminal part of ACB system where the conditioned air is discharged into the occupied place. It is normally mounted inside or below ceiling and consists of a primary air plenum, nozzles, a mixing chamber and a heat exchanger. The working principle is depicted in Figure 1. After being processed by air handling unit and dehumidifier, the conditioned air is sent to terminal unit of ACB system. Firstly, it is forced into dozens of small nozzles by high pressure from the primary air plenum and ends up with high speed primary air. An entrainment effect is generated by this high speed fluid. The indoor air is induced into the beam through a water-air cooling heat exchanger, namely secondary air. The primary air and secondary air mix together in the mixing chamber and serve as the supply air for the occupied zone as a whole.

In modern society, people spend most of their time in the artificial environment. Increasing diseases such as sick building syndrome make people suffer so much because of the poor thermal environment and air quality[3]. Studies show that many factors including air distribution, temperature, heat sources and human activities all have significant effects on the thermal comfort. Air distribution is one of the most important factors. Kosonen et al. [4] discussed the influence of thermal load strength and position on airflow pattern of chilled beams in a test room. The experiment result showed that the maximum velocity increased as the rising of heat load. However, the location of the thermal load did not significantly affect the maximum velocity. The un-uniform air distribution is easily to cause turbulence, which has high risk to induce draught sensation. Jan et al. [5] examined the velocities around the chilled beams based on a thermistor anemometer system. It was demonstrated that the airflow generated by a chilled beams exhibits strong fluctuations which may result in discomfort draught. The airflow pattern was extremely unstable and sensitive to the presence of heat sources. Cao et al. [6, 7] utilized an original method, particle image velocimetry (PIV), to trace the path of the airflow. The attached plane jet moved in a turbulent way regardless of the Reynolds number.

The previous studies for air distribution are mainly focused on the occupied places. However, little attention is paid to air distribution in the vicinity of ACB terminal unit, which is exactly the air source for the indoor room. The non-uniformity of the airflow has a huge potential to cause uncomfortable draught for occupants. In order to fill the gap, air distribution for a two-way discharge ACB terminal unit is investigated in this study.

Computational Fluid Dynamics (CFD) technology has been widely used in field of fluid analysis as a useful tool to predict the air movement since 1970s. Besides, it also plays an important part in visualizing the flows in 3D spaces [8-10].The airflow in the room is described mathematically by a set of couplet differential equations known as the Navier-Stokes equations and the procedure of solving these equations by a numerical method is called computational fluid dynamics [11]. With the rapid development of ACB, CFD is also applied as powerful tool by some scientists studying chilled beams. Hannu et al. [12, 13] characterized the air distribution in office environment that was ventilated by ACB under summer, spring, autumn and winter cases. The layout of ACB was in an asymmetric way and the corresponding CFD models were set up to simulate the flow pattern. The conclusions were drawn that the heat sources provide a notable impact on the air distribution. More importantly, a large circulation occurred because of asymmetric layout of chilled beams. Einberg et al. [14] predicted the velocity close to an air diffuser by means of CFD simulations and laboratory experiments. The conclusion was that the airflow near the diffuser was a complex combination of jet flow and flow from a displacement diffuser. As the flow pattern near the nozzle duct were too complicated to be modelled in practical CFD simulations, a simplified CFD model was developed for nozzle duct diffuser by Hannu. However, he found that the simplified model could not be extensively applicable[15]. Therefore, a CFD model with original scale was set up in this paper to investigate the air distribution of ACB.

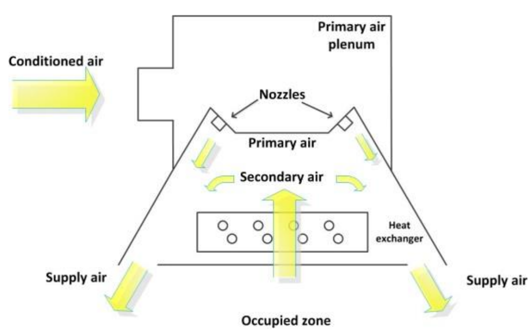

Figure 1 Schematic diagram of ACB

2 CFD modeling of ACB

2.1 CFD modeling of ACB

2.1.1 Computational grid

A three dimensional ACB model was set up by Computational Fluid Dynamics (CFD) of Ansys edition 12.0. As ACB is symmetry from left to right side, only half of ACB was built to simplify the calculation. The computational grid is formed by tetrahedron cells (Figure 2) because of the complex structrue of ACB. The minimum element quality is 0.213 and the elements number is 2595478.

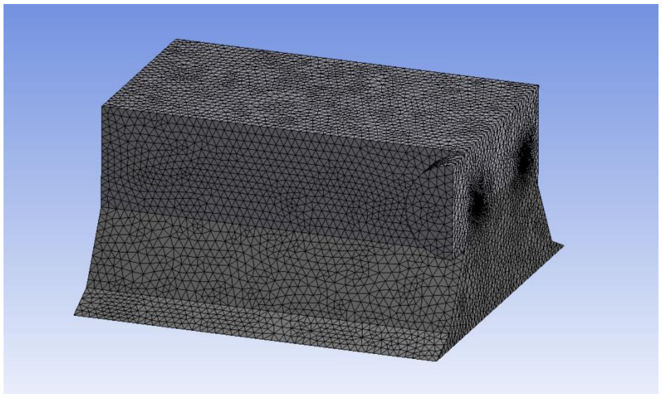

Figure 2: CFD model mesh diagram of ACB

2.1.2 Boundary conditions

The Reynolds number for inlet is estimated from the equation:

$$\mathrm{Re} = \mathrm{vL}/\nu = \rho \mathrm{vL}/\mu$$

Where v is the velocity of fluid, L is the characteristic length, ν is the kinematic viscosity, μ is the dynamic viscosity of the fluid, ρ is the density of the fluid.

For fully developed pipe flow the turbulence intensity at the core, the turbulence intensity can be estimated from the equation:

$$\mathrm{I} = 0.16 * (\mathrm{Re})^{-\frac{1}{8}}$$

where Re is the Reynolds number based on the pipe hydraulic diameter.

As the Reynolds number is around 1.93×10^{4}, the air flow is considered to be constant and incompressible. The turbulence intensity is estimated as 5%.The inlet pressure is set to be 30Pa. The outlet pressure and secondary air inlet pressure is 0 Pa. Other prameters are all default values.

2.1.3 Solver settings

The standard k-ε model, which is widely used in fluid dynamics, is adoped as the turbulence model. The SIMPLEC algorithm is used for pressure-velocity coupling. Second order upwind disccretization scheme is applied in the solving momenterm, turbulence energy and turbulent dissipation rate terms.

The basic governing equations for incompressible fluid are:

(1) Mass conservation equation(Continuity equation)

$$\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}+\frac{\partial w}{\partial z}=0 \tag{1}$$

where, u, v and w are the velocities of x, y and z direction.

(2) Momentum conservation equation

$$\begin{cases} \frac{\partial u}{\partial t}+\text{div}(u\,\text{U})=\text{div}(\upsilon \text{grad}u)\text{-}\frac{1}{\rho}\frac{\partial p}{\partial x} \\ \frac{\partial v}{\partial t}+\text{div}(v\,\text{U})=\text{div}(\upsilon \text{grad}uv)\text{-}\frac{1}{\rho}\frac{\partial p}{\partial y} \\ \frac{\partial w}{\partial t}+\text{div}(w\,\text{U})=\text{div}(\upsilon \text{grad}w)\text{-}\frac{1}{\rho}\frac{\partial p}{\partial z} \end{cases} \tag{2}$$

where p is the pressure of the air; υ denotes the kinematic viscosity; ρ is the density.

(3)The k-ε model equations:

$$\rho\frac{\text{dk}}{dt}=\frac{\partial}{\partial x_i}\left[\left(\mu+\frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_i}\right]+G_k+G_b-\rho\varepsilon-Y_M \tag{3}$$

$$\rho\frac{\text{d}\varepsilon}{dt}=\frac{\partial}{\partial x_i}\left[\left(\mu+\frac{\mu_t}{\sigma_\varepsilon}\right)\frac{\partial \varepsilon}{\partial x_i}\right]+G_{1\varepsilon}\frac{\varepsilon}{\text{k}}(\text{G}_k+G_{3\varepsilon}G_b)-G_{2\varepsilon}\rho\frac{\varepsilon^2}{\text{k}} \tag{4}$$

where k is turbulence energy; ε is turbulent dissipation rate; G_k is turbulent kinetic energy caused by mean velocity gradient; G_b is the turbulent kinetic energy caused by buoyancy; Y_M denotes effects of compressible turbulent flow pulsation to the total turbulent dissipation.; μ is kinematic viscosity; others are considered constants in CFD.

2.2 CFD model calibration by experiments

This CFD model is verified by experiments in a 7.3m*3.3m*2.5m mock up room (Figure 3). An ACB (Figure 4) with a dimension of 1200mm*400mm*158mm is equipped latitudinal in the middle of the room. 30 nozzles with diameter of 21.5mm are distributed on each side of this ACB. The velocities around the nozzles are tested by a hot wire air velocity transducer. The full scale is 10m/s with accuracy of ±0.5%.

Figure 3: Thermal room

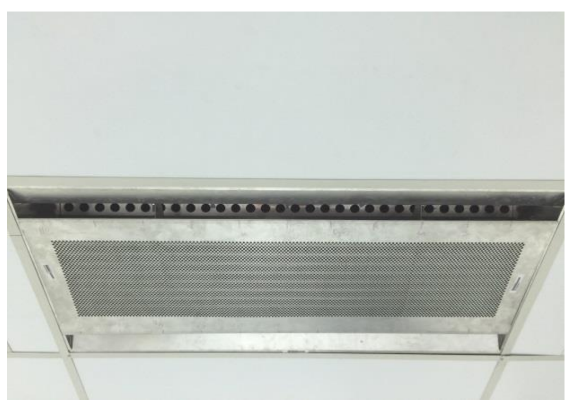

Figure 4: ACB terminal unit

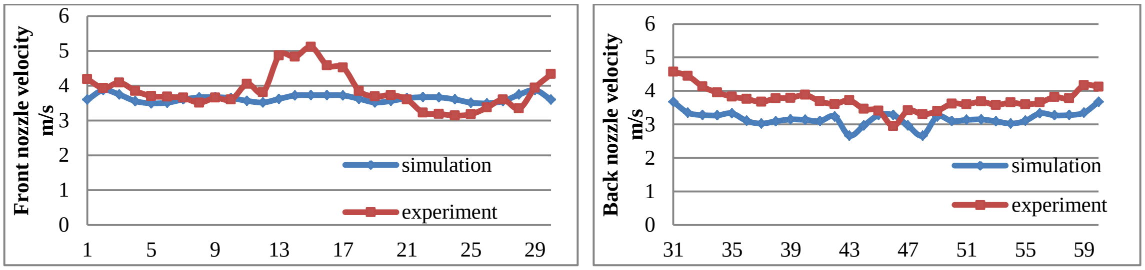

Figure 5: Nozzle velocity comparison between simulation and experiment

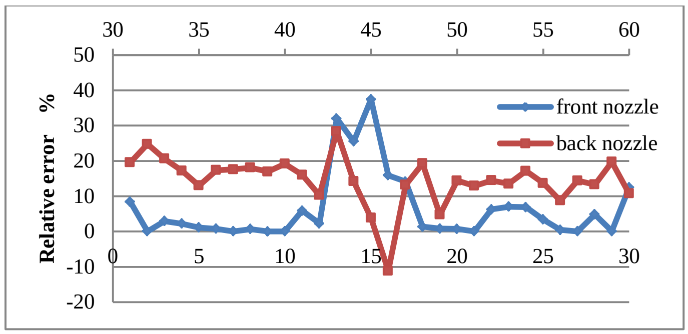

Figure 6: Relative error between experiment and simulation

Assume the inlet side is the back side of ACB, where nozzles are sequenced from 31 to 60. On the contrary, points 1-30 are nozzles located in the opposite side of inlet. The nozzle velocity comparison between simulation and experiment is show in Figure 5. For front nozzles, the velocities match quite well with the simulation velocities except for a few middle points. The overall trend for experiment is similar for back nozzles compared with the simulation results. The velocity magnitude difference between experiment and simulations may be caused by simulation and testing errors. The relative error between experiment and simulation are also calculated (Figure 6). Most relative errors are within 20%, which can be acceptable for CFD simulation.

3 Results and analysis

3.1Simulation results

The simulation air streamlines are shown in Figure 7, from which we can see the air velocity distribution is quite fluctuant in the primary air plenum. It can be drawn from the line charts (Figure 8) that, a huge velocity difference exists between back nozzles and front nozzles. For back nozzles (nozzles 31-60), the velocities reach the bottom at the middle area which is close to inlet area. However, the velocities on the two edges are slightly large. In terms of front nozzles (nozzles 1-30), the velocities have higher magnitudes at the middle and sides area. The maximum velocity difference is 0.957m/s. As a result of the velocity un-uniformity between front and back nozzles, the air flow rates discharging from two outlets are also different, which may have an effect on the ventilation in the room.

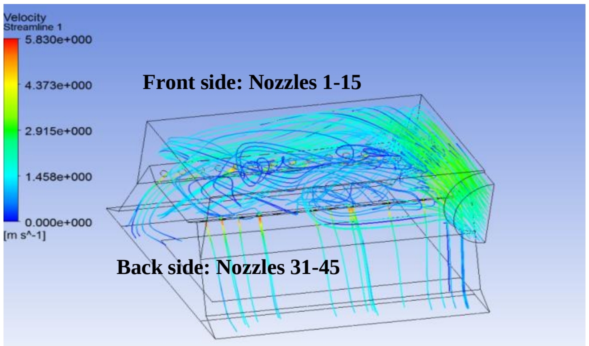

Figure 7: ACB simulation streamlines

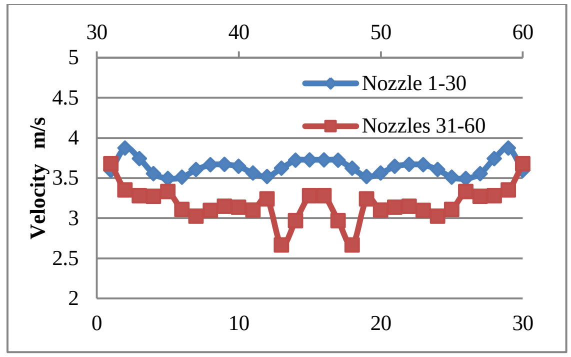

Figure 8: Simulation velocities for nozzle diameter 21.5mm under 30Pa

In order to investigate the effects of nozzle diameter and inlet pressure on this velocity un-uniformity between front and back nozzles, the CFD model is applied to simulate the velocities for nozzle diameter21.5mm, 15mm and 9mm under inlet pressure 30 Pa, 60 Pa and 90 Pa, respectively. The simulation results are depicted in Figure 9, 10 and 11.

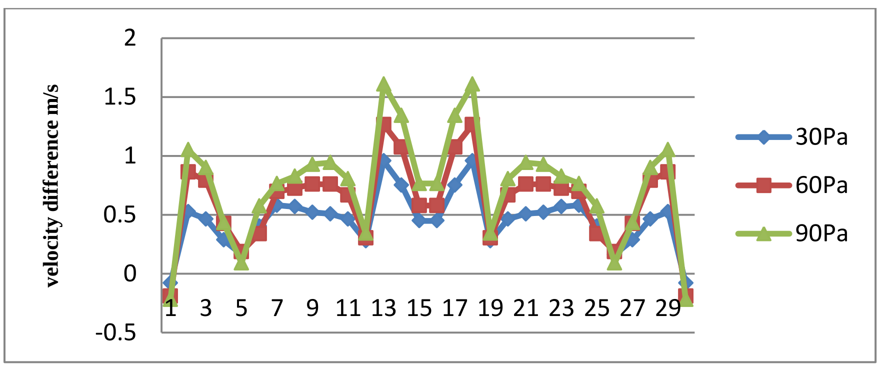

Figure 9: Velocities difference for nozzle diameter 21.5mm under 30Pa, 60Spa, 90Pa

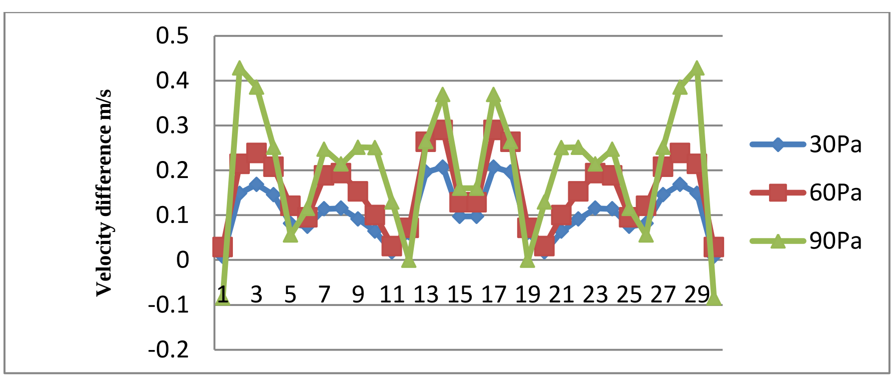

Figure 10: Velocities difference for nozzle diameter 15mm under 30Pa, 60Spa, 90Pa

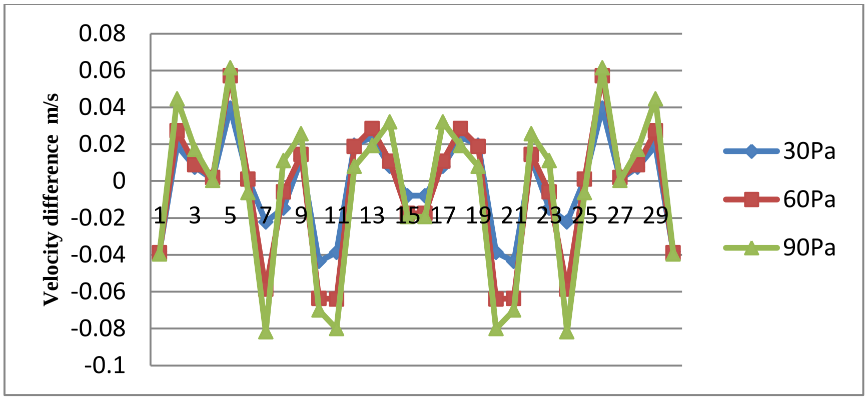

Figure 11: Velocities difference for nozzle diameter 9mm under 30Pa, 60Spa, 90Pa

As is shown in the Figure 9, 10 and 11, with a decrease of nozzle diameter, the velocity difference reduces. When the nozzle is small enough, the velocity differences are small enough to be neglected. Moreover, for the same nozzle diameter, the higher the inlet pressure, the bigger of the gap between two lines nozzles. The maximum velocity difference magnitude is depicted in Table 1.

Table 1: maximum velocity difference magnitude

Inlet Pressure / Diameter	30 Pa	60 Pa	90 Pa
9mm	0.044m/s	0.064 m/s	0.082 m/s
15mm	0.207 m/s	0.290 m/s	0.428m/s
21.5mm	0.957 m/s	1.265m/s	1.609 m/s

3.2A improvement for un-uniformity of nozzle velocity

In order to solve this unpleasant un-uniformity, several solutions have been proposed. Only few have been proved to be feasible through validation of CFD. One of the strategies is to add a board (200*70*2mm) in the middle of the primary air plenum as is shown in Figure 12.

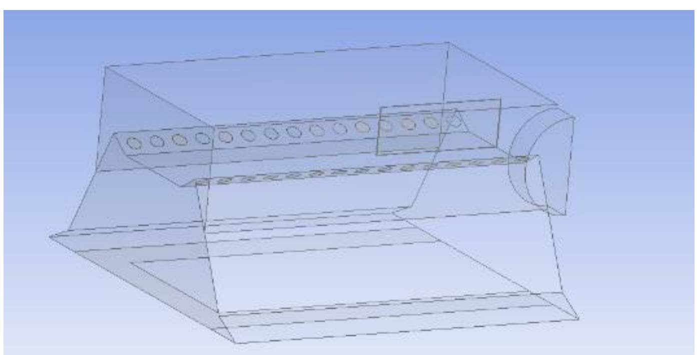

Figure 1: Modified ACB

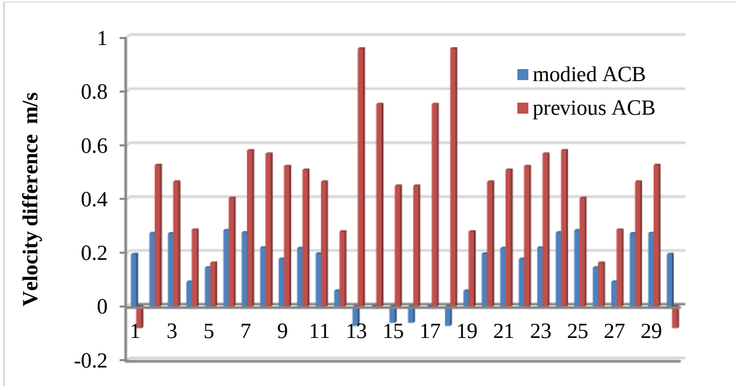

Figure 2: Velocity differences between front and back nozzles

As is depicted in the **Error! Reference source not found.**, the velocities differences between front side and back nozzles are smaller after being modified. The maximum velocities between front and back nozzles are 0.378m/s, which is obviously smaller than maximum difference 0.957 m/s of the original ACB. It can be concluded that adding a board in the middle will significantly improve the uniformity of velocities.

4 Conclusions

From the results of simulation and experiments of air distribution for ACB nozzles, the conclusion can be drawn that the un-uniformity is significant when the nozzle diameter is large. Moreover, the rise of the inlet pressure will aggravate this un-uniformity. The smallest velocities are located close to inlet of back side, while the largest velocities are distributed in the nozzles of front side and the two verges. The pressure around the opposite side of inlet and edge area is comparatively larger which may leads to the high speed in those areas. However, this troublesome un-uniformity will be eliminated when the nozzle is small enough. When nozzles are small, the air flow velocity is primarily restricted by the pore size. However, for the nozzles owning a large pore size, the air velocity will mainly depend on the pressure for the nozzle. Therefore, as we design the ACB systems, high attention on the nozzle diameter should be paid to prevent the un-uniformity air flow when the nozzles and the inlet pressure are large.

In addition, a solution of adding a board in the middle is proposed and validated by CFD simulation. With the help of this board, the air will be discharged evenly from the nozzles. Through a complex process, the air will be distributed to all of the nozzles in a more balanced way. The corresponding experiments will be conducted later. The study of air distribution inside ACB is of great significance to reduce the un-uniformity of air flow form the source and provide occupants a pleasant indoor environment with little chance of draught problem.

Reference

[1] C. Chen, W. Cai, Y. Wang, and C. Lin, "Performance comparison of heat exchangers with different circuitry arrangements for active chilled beam applications," *Energy and Buildings*, vol. 79, pp. 164-172, 8// 2014.

[2] ASHRAE Standard, "113 (2005)“Method of testing for room air diffusion” American Society of Heating, Refrigerating and Air-Conditioning Engineers," ed: Inc.

[3] G. Cao, H. Awbi, R. Yao, Y. Fan, K. Sirén, R. Kosonen, *et al.*, "A review of the performance of different ventilation and airflow distribution systems in buildings," *Building and Environment*, vol. 73, pp. 171-186, 3// 2014.

[4] R. Kosonen, A. Melikov, L. Bozkhov, and B. Yordanova, "Impact of heat load distribution and strength on airflow pattern in rooms with exposed chilled beams," in *Proceedings of Roomvent*, 2007.

[5] J. Fredriksson, M. Sandberg, and B. Moshfegh, "Experimental investigation of the velocity field and airflow pattern generated by cooling ceiling beams," *Building and Environment,* vol. 36, pp. 891-899, 8// 2001.
[6] G. Cao, "Modelling the attached plane jet in a room," 2009.
[7] G. Cao, M. Sivukari, J. Kurnitski, M. Ruponen, and O. Seppänen, "Particle Image Velocimetry (PIV) application in the measurement of indoor air distribution by an active chilled beam," *Building and Environment,* vol. 45, pp. 1932-1940, 9// 2010.
[8] J. J. Martinez-Almansa, A. Fernandez-Gutierrez, L. Parras, and C. del Pino, "Numerical and experimental study of a HVAC wall diffuser," *Building and Environment,* vol. 80, pp. 1-10, 10// 2014.
[9] J. D. Posner, C. R. Buchanan, and D. Dunn-Rankin, "Measurement and prediction of indoor air flow in a model room," *Energy and Buildings,* vol. 35, pp. 515-526, 6// 2003.
[10] G. Zhang, S. Morsing, B. Bjerg, K. Svidt, and J. S. Strøm, "Test Room for Validation of Airflow Patterns estimated by Computational Fluid Dynamics," *Journal of Agricultural Engineering Research,* vol. 76, pp. 141-148, 6// 2000.
[11] P. V. Nielsen, "Fifty years of CFD for room air distribution," *Building and Environment*, vol. 91, pp. 78-90, 9// 2015.
[12] H. Koskela, H. Häggblom, R. Kosonen, and M. Ruponen, "Air distribution in office environment with asymmetric workstation layout using chilled beams," *Building and Environment*, vol. 45, pp. 1923-1931, 9// 2010.
[13] H. Koskela, H. Häggblom, R. Kosonen, and M. Ruponen, "Flow pattern and thermal comfort in office environment with active chilled beams," *HVAC&R Research,* vol. 18, pp. 723-736, 2012.
[14] H. Awbi, "Application of computational fluid dynamics in room ventilation," *Building and environment,* vol. 24, pp. 73-84, 1989.
[15] M. Cehlin and B. Moshfegh, "Numerical and experimental investigations of air flow and temperature patterns of a low velocity diffuser," in *Proceedings of the ninth international conference on indoor air quality and climate*, 2002, pp. 765-70.