Air distribution and thermal comfort for active chilled beam systems

Wu, Bingjie

2018

Wu, B. (2018). Air distribution and thermal comfort for active chilled beam systems.
Doctoral thesis, Nanyang Technological University, Singapore.

<https://hdl.handle.net/10356/85020>

<https://doi.org/10.32657/10220/46679>

**NANYANG
TECHNOLOGICAL
UNIVERSITY**

SINGAPORE

**AIR DISTRIBUTION AND THERMAL COMFORT FOR
ACTIVE CHILLED BEAM SYSTEMS**

WU BINGJIE

**Interdisciplinary Graduate School
Energy Research Institute @ NTU**

2018

**AIR DISTRIBUTION AND THERMAL COMFORT FOR
ACTIVE CHILLED BEAM SYSTEMS**

WU BINGJIE

INTERDISCIPLINARY GRADUATE SCHOOL

A thesis submitted to the Nanyang Technological University in
partial fulfilment of the requirement for the degree of Doctor of
Philosophy

2018

Statement of Originality

I hereby certify that the work embodied in this thesis is the result of original research and has not been submitted for a higher degree to any other University or Institution.

15th, Aug, 2018

Wu Bingjie

.....

Date

.....

Input Name Here

Abstract

During the past few decades, the active chilled beam (ACB) system has become increasingly prevalent worldwide as a promising air conditioning and mechanical ventilation (ACMV) system. It is acknowledged that ACB systems have the advantages of providing high energy efficiency, good thermal comfort, and satisfied noise control with a lower space requirement. In modern society, the thermal discomfort and excessive energy consumption are the two major concerns for ACMV systems. People, especially in developed countries, spend substantial of their time in air-conditioned rooms. The poor air quality and inferior thermal condition incur plenty of uncomfortable sensations and even give rise to sicknesses such as sick building syndrome. One of the most common and serious problems is sensation of draught, which is mainly caused by the unpleasant cooling of air movement. Therefore, it is significantly important to evaluate the air distribution and thermal comfort for ACB systems. However, the previous studies mainly focused on cooling capacity, energy-saving effect and mechanical optimization of ACB systems. The studies on air distribution and thermal comfort are still inadequate to provide a comprehensive perspective on ACB systems.

In order to fulfil the gaps, the airflow pattern is experimentally and numerically studied for ACB systems. Through experiments, an un-uniform air distribution is found near the nozzles in ACB terminals and this found distribution has a great chance to cause uncomfortable feeling such as draught in the occupied zone. A three-dimensional computational fluid dynamics (CFD) model for ACB terminal is built and validated by particular experiments. A proper strategy to balance the air distribution is proposed and validated by CFD simulation. Besides, the velocity contours near the ceilings are captured and the characteristics of Coandă effect, self-similarity, and turbulence intensity for air jet are discussed under various air supply temperatures and pressures.

In addition, the effects of the heat sources' configuration and strength on thermal comfort are experimentally investigated in a mock-up room with ACB systems. The full-scale

distributions of air speed, temperature and humidity are mapped. Based on the data, the thermal comfort is comprehensively evaluated with indices of air diffusion performance index, predicted mean vote, draught rate and vertical air temperature difference. The uniformities of air speed, temperature and humidity are investigated in terms of standard deviation. Practical schemes are provided on how to set up sensors and controllers of ACB systems under various operation conditions.

By making full use of the obtained characteristics of air distribution and thermal comfort, a model-based genetic algorithm is employed to achieve the minimal energy consumption of ACB systems while maintain the indoor thermal comfort for the occupied zone. The optimal energy consumption and thermal comfort are found by regulating the primary airflow rate and water flow rate. The optimization results show better performances compared with the original experimental results.

Acknowledgements

First and foremost, I would like to express my sincere gratitude to my supervisors, Professor Cai Wenjian, co-supervisor, Professor Anutosh Chakraborty and mentor Professor Wen Changyun, for their tremendous support and invaluable guidance throughout the course of my research work.

I also would like to thank the Energy Research Institute @NTU (ERI@N), Interdisciplinary Graduate School, NTU, for their financial support for my Ph.D study.

I particularly thank my colleagues and seniors Mr. Ji Ke, Mr. Zhai Deqing, Dr. Chen Haoran, Dr. Chen Can, Dr. Lin Chen, Dr. Shen Suping, Mr. Ou Xianhua, Dr Wu Qiong, Dr Wang Xinli in the Process Instrumentation Laboratory for their patient assistance through my research work. They always are willing to sacrifice their time to give me instructions and pull me out when I am in trouble.

Special thanks are given to my parents and my boyfriend for their steadfast support and companionship.

Table of Contents

Abstract.....	I
Acknowledgements	III
Table of Contents	V
List of Tables	IX
List of Figures.....	XI
List of Symbols	XV
Chapter 1 Introduction.....	1
1.1 Background	1
1.2 Overview of ACB systems.....	4
1.3 Motivations and objectives	6
1.4 Major contributions.....	7
1.5 Organization of the thesis	9
Chapter 2 Literature Review	11
2.1 Introduction.....	11
2.2 Mechanical design of ACB systems	11
2.3 Models for ACB systems	12
2.4 Air distribution for ACB systems	13
2.5 Thermal comfort for ACB systems.....	18
2.6 CFD applications	23
2.7 Summary	24
Chapter 3 CFD simulation for airflow patterns.....	25
3.1 Introduction.....	25
3.2 CFD modeling of ACB	25

3.2.1 Computational grid	25
3.2.2 Boundary conditions	27
3.2.3 Solver settings.....	27
3.3 CFD model validation.....	27
3.4 Results and analysis	30
3.4.1 Simulation results	30
3.4.2 An optimized strategy to improve un-uniformity of nozzle velocity	33
3.5 Summary	34
Chapter 4 Experimental investigation on airflow patterns.....	37
4.1 Introduction.....	37
4.2 Experimental setup.....	38
4.2.1 The ACB terminal.....	38
4.2.2 The test room and measurement point distribution	38
4.2.3 Measurement devices.....	39
4.2.4 Measurement procedures	40
4.3 Results.....	42
4.3.1 Velocity contours.....	42
4.3.2 Velocity profiles along streamwise direction	44
4.3.3 Velocity profiles along vertical direction	46
4.4 Discussion	48
4.4.1 Coandă effect	48
4.4.2 Self-similarity of the air jet.....	48
4.4.3 Turbulence intensity	51
4.5 Summary	53
Chapter 5 Heat source effects on thermal comfort.....	55

5.1 Introduction.....	55
5.2 Experimental setup.....	56
5.2.1 The test room and measurement point distribution	56
5.2.2 Measurement device	57
5.2.3 Measurement procedures	58
5.3 Results.....	61
5.3.1 Air speed distribution	62
5.3.2 Air temperature distribution	64
5.3.3 Air humidity distribution	66
5.3.4 Uniformity	68
5.3.5 ADPI.....	70
5.3.6 Predicted mean vote (PMV)	72
5.3.7 Draught rate & Vertical air temperature difference.....	74
5.4 Summary	74
Chapter 6 Optimization for the ACB systems.....	77
6.1 Introduction.....	77
6.2 Model development	77
6.2.1 Chiller model	77
6.2.2 Model of fan and pump.....	79
6.2.3 Heat transfer model.....	79
6.2.4 Thermal comfort model	80
6.3 Global optimization strategy.....	81
6.3.1 Objective function	81
6.3.2 Constrains	81
6.3.3 Genetic algorithm optimization strategy	83

6.4 Experimental setup and model validation.....	86
6.4.1 Experimental setup	86
6.4.2 Model validation.....	88
6.5 Optimization results	92
6.6 Summary	94
Chapter 7 Conclusions and future work.....	95
7.1 Conclusions.....	95
7.2 Future work.....	96
References	101

List of Tables

Table 2.1 Seven-point thermal sensation scale	20
Table 2.2 Categories of thermal environment.....	21
Table 3.1 Maximum velocity differences	33
Table 4.1 Sensor specifications.....	40
Table 4.2 Measurement conditions	41
Table 5.1 Sensor specification	58
Table 5.2 Experimental parameters	60
Table 5.3 Experimental uncertainty analysis	60
Table 5.4 Thermal comfort performance	74
Table 6.1 GA settings	84
Table 6.2 Components nominal capacities	88
Table 6.3 Accuracy of proposed models.....	91
Table 6.4 Three original cases	92
Table 6.5 Effects of weight factor λ	94

List of Figures

Figure 1.1 Typical ACMV systems: (a) FCU system; (b) VAV system; (c) ACB system; (d) PCB system; (e) Radiant cooling system	3
Figure 1.2 Structure of ACB	4
Figure 1.3 Schematic diagram of ACB	5
Figure 2.1 Typical air distribution: (a) MV; (b) DV	14
Figure 2.2 Coandă effect	15
Figure 2.3 Self-similarity	15
Figure 3.1 CFD model mesh	26
Figure 3.2 Velocity comparison for two mesh models for nozzles 1-15	26
Figure 3.3 The ACB terminal	28
Figure 3.4 Experimental results with error bars	28
Figure 3.5 Nozzle velocity comparison between simulation and experiment: (a) front nozzles; (b) back nozzles	29
Figure 3.6 Relative error between experiment and simulation	29
Figure 3.7 ACB simulation streamlines for nozzle diameter 21.5mm under 30 Pa	30
Figure 3.8 Simulation velocities for nozzle diameter 21.5mm under 30 Pa	31
Figure 3.9 Velocity differences for nozzle diameter 21.5mm under 30 Pa, 60 Pa, 90 Pa	31
Figure 3.10 Velocity differences for nozzle diameter 15mm under 30 Pa, 60 Pa, 90 Pa	32
Figure 3.11 Velocity differences for nozzle diameter 9mm under 30 Pa, 60 Pa, 90 Pa	32
Figure 3.12 Modified ACB terminal	33
Figure 3.13 Velocity differences between front and back nozzles	34
Figure 4.1 (a) ACB terminal unit; (b) Thermal isolated room	38
Figure 4.2 Measurement point distribution	39
Figure 4.3 (a) Velocity contours for isothermal conditions (cases 1-3); (b) Velocity contours for non-isothermal conditions (cases 1, 4, 5)	43

Figure 4.4. (a) Velocity profiles along streamwise direction for isothermal conditions (cases 1-3); (b) Velocity profiles along streamwise direction for isothermal conditions (cases 1-3) at $y=0.15\text{m}$; (c) Velocity profiles along streamwise direction for non-isothermal conditions (cases 1, 4, 5); (d) Velocity profiles along streamwise direction for non-isothermal conditions (cases 1, 4, 5) at $y=0.15\text{m}$; where y is vertical distance from the ceiling.....	45
Figure 4.5 (a) Velocity profiles along vertical direction for isothermal conditions (cases 1-3) as well as the change in vertical peak velocity; (b) Velocity profiles along vertical direction non-isothermal conditions (cases 1, 4, 5) as well as the change in vertical peak velocity.....	47
Figure 4.6 (a) Relationship of y and normalized velocity (u/U_m) for isothermal conditions (cases 1-3); (b) Relationship of y and normalized velocity (u/U_m) for non-isothermal conditions (cases 1, 4, 5); where x is the horizontal distance from the outlet of ACB	50
Figure 4.7 Model comparison	51
Figure 4.8 Mean absolute errors of two models	51
Figure 4.9 (a) Turbulence intensity for isothermal conditions (cases 1-3); (b) Turbulence intensity for non-isothermal conditions (cases 1, 4, 5); where y is the vertical distance from the ceiling	52
Figure 5.1 The test room: 1- ACB; 2 – heating panels; 3- mobile robot	57
Figure 5.2 Tesingt point distribution	57
Figure 5.3 Configurations of heating panels: (a) configuration I ; (b) configuration II ; (c) configuration III	61
Figure 5.4 (a)-(f): Air speed distribution for cases 1-6, respectively.....	63
Figure 5.5 (a)-(f): Air temperature distribution for cases 1-6.....	65
Figure 5.6 (a)-(f): Absolute humidity distribution for cases 1-6	67
Figure 5.7 (a): Uniformities of air temperature, speed and absolute humidity; (b):Uniformities of air absolute humidity for each height; (c):Uniformities of air temperatures for each height;.....	69
Figure 5.8 (a)-(e): ADPI results for cases 2-6.....	71
Figure 5.9 (a)-(e): PMV values for cases 2-6	73

Figure 6.1 Flow chart of the optimization strategy	85
Figure 6.2 Schematic diagram of the ACB system	86
Figure 6.3 Experimental facilities for the ACB system: 1: AHU; 2: Fan; 3: Water tank; 4: Water pump for chiller 1; 5: Water pump for chiller 2; 6: Water pump for coil in ACB terminal; 7: Compressor for chiller 1; 8: Compressor for chiller 2; 9: Evaporator for chiller 1; 10: Evaporator for chiller 2; 11: Condenser for chiller 1; 12: Condenser for chiller 2; 13: Control cabinet	87
Figure 6.4 LabView program.....	88
Figure 6.5 Model validation for the fan energy model	89
Figure 6.6 Model validation for pump energy model	89
Figure 6.7 Model validation for chiller 1 energy model	90
Figure 6.8 Model validation for chiller 2 energy model	90
Figure 6.9 Model validation for heat transfer model of coil in ACB terminal	91
Figure 6.10 Model validation for maximum air velocity in the occupied zone.....	91
Figure 6.11 The original and optimization results for energy consumption.....	92
Figure 6.12 The original and optimization results for PPD	93
Figure 6.13 Effects of weight factor λ	93

List of Symbols

Letters

a	coefficient
A	constant coefficient
Ar	Archimedes number
c	specific heat of chilled water
c_1	coefficient
c_2	coefficient
e	error
E	energy (W)
erf	error function
exp	exponential function
f_{cl}	clothing surface area factor
g	gravitational acceleration
h	enthalpy (kJ/kg)
h_c	convective heat transfer coefficient (W/(m ² ·K))
H	room height (m)
I_{cl}	clothing insulation (m ² ·K/W)
K	heat transfer coefficient (W/(m ² ·K))
L	characteristic length (m)
m	constant coefficient
$\dot{m}$	mass flow rate (kg/s)
M	metabolic rate (W/m ²)
MAE	mean absolute errors
n_1	coefficient
n_2	coefficient
N	total testing points
p	pressure (Pa)
p_a	water vapor partial pressure
P_w	water vapor pressure
P_{ws}	saturated water vapor pressure (hPa)

PLR	part load ratio
Q	heat transfer capacity (W)
R_w	constant coefficient
Re	Reynolds number
RH	relative humidity (%)
S_v	standard deviation
t	temperature (°C)
t_{cl}	clothing surface temperature
T	temperature (°C)
ΔT	temperature difference (°C)
TI	turbulence intensity
T_n	constant coefficient
u	velocity (m/s)
u_0	initial velocity of the supply air (m/s)
U_m	maximum velocity (m/s)
v	velocity (m/s)
$\bar{v}$	average velocity
$\dot{V}$	volume flow rate (m ³ /h)
W	effective mechanical power (W/m ²)
x	horizontal distance from the outlet of ACB (m)
x_i	testing data
y	local vertical position (m)

Subscripts

a	air
avg	average
b	buoyancy
c	chiller
cur	current
ed	effective draught
f	fan
i	counting number
nom	nominal
p	pump
req	requirement
RH	absolute humidity
w	water

Greek symbols

β	volume expansion coefficient
ρ	density (kg/m ³)
μ	dynamic viscosity
λ	weigh factor
η	$y/y_{1/2}$
ν	kinematic viscosity
ω	velocity of water

Abbreviations

ACB	active chilled beam
ACMV	air conditioning and mechanical ventilation
AHU	air handling unit
ADPI	air diffusion performance index
ASHRAE	American Society of Heating, Refrigeration and Air-Conditioning Engineer
CFD	computational fluid dynamics
DR	draught rate
DOAS	dedicated outdoor air system
DV	displacement ventilation
FCU	fan coil unit
GA	genetic algorithm
GHG	greenhouse gas
IEQ	indoor environment quality
IR	induction ratio
SBS	sick building syndrome
MV	mixing ventilation
NC	noise criterion
NTU	number of transfer units
PMV	predicted mean vote
PCB	passive chilled beam
PPD	predicted percentage dissatisfied
VAV	variable air volume
VATD	vertical air temperature difference

Chapter 1 Introduction

1.1 Background

Air conditioning and mechanical ventilation (ACMV) systems have gradually play a crucial role in human being's life ever since the first modern electrical air conditioning unit was introduced in 1902 [1]. ACMV systems are widely utilized in industries, commercial buildings, households, transportations vehicles, and so on. Initially, ACMV systems were designed to maintain a comfort environment and ensure ventilation for occupants. However, thermal comfort has become a major challenge nowadays. Some diseases such as sick building syndrome (SBS) arise due to poor indoor thermal environment and air quality. The reasons behind are usually inappropriate operations of ACMV systems and inferior system designs. In urbanized cities, people spend up to 90% of their time indoors [2]. More attention has to be paid on thermal comfort and air quality that are governed by the ACMV systems. The other challenge for ACMV systems is excessive energy consumption. The energy shortage and increasing emissions of greenhouse gas (GHG) have raised many concerns worldwide [3]. In 2010, buildings occupied 32% of total global final energy use, 33% of black carbon emissions, and 19% of energy-related GHG emissions [4]. With rapid economic development in China, the building energy consumption in China was increased from 10% of the national total energy in the late 1970s to more than 25% in 2006, and is expected to be 35% by 2020 [5-7]. According to Singapore Energy Statistics, commercial buildings occupied 36.8% of entire electricity consumption for Singapore in 2016 [8]. Cooling and ventilation account for around 60% and 10% of the building energy consumption respectively [9]. It can be seen that ACMV systems consume a majority part of energy which gives rise to substantial GHG emissions. There is a huge potential to save more energy for ACMV systems not only from system design, but also from the control and operation aspect.

Several typical cooling ACMV systems are fan coil unit (FCU) systems, variable air

volume (VAV) systems, radiant cooling systems, and chilled beam systems. The chilled beam systems can be further divided into active chilled beam (ACB) systems and passive chilled beam (PCB) systems (Figure 1.1). The FCU terminal normally consists of a cooling or heating coil and a fan. FCU systems have high level of flexibility in terms of subdivision and rearrangement of space. However, FCU systems seldom use outside air for free cooling, which makes the system less efficient in spring and autumn [10]. As FCU fans are installed near the occupied zone, the noise is higher than the other systems. VAV systems normally provide airflow with constant temperatures. The temperature in the occupied zone is adjusted by varying the airflow rate, which leads to higher risk of uncomfortable draught. Sometimes, the occupied zone will be overcooled to meet the minimum ventilation requirements. In order to solve this problem, a preheat device will be attached for VAV systems to maintain the room temperature, which causes unnecessary energy waste. FCU and VAV systems reject heat mainly by forced convection. In contrast, radiant cooling systems exchange the heat with surrounding environment through radiation and natural convection. It is commonly believed that the radiant cooling systems have better thermal comfort performance in comparison to other all-air system due to the reduced air movement and draught problems [11, 12]. However, as there is no ventilation, extra ventilation systems need to be added to ensure the air quality for radiant panel systems. PCB systems provide cooling effects by natural convection and radiation and have no direct air supply, while ACB systems are able to offer cooling or heating function as well as fresh airflow to the occupied space through forced and natural convection and radiation. Due to forced convection, ACB achieves cooling densities about twice of those of PCB [13]. Dedicated outdoor air system (DOAS) systems including dehumidifiers are often combined with ACB systems. ACB systems plus DOAS systems are superior to VAV systems in terms of energy efficiency, first cost, air quality, etc [14-17]. Due to their compelling benefits with respect to energy efficiency, thermal comfort, and noise control with less space requirement, ACB systems have become reliable ACMV solutions and have been increasingly popular in not only Europe, but also North America and Asia [18]. They are widely utilized in a variety of commercial buildings, schools, laboratories and hospitals [19].

Figure 1.1 Typical ACMV systems: (a) FCU system; (b) VAV system; (c) ACB system; (d) PCB system; (e) Radiant cooling system

1.2 Overview of ACB systems

ACB is the terminal part of an 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, as it is shown in Figure 1.2. ACB system is an air-water ACMV system and has two sources for supply air: primary air and induced air. The primary air is forced fresh air that can be conditioned and dehumidified with a DOAS. The induced air is entrained by the primary air from the occupied zone and cooled by a heat exchanger inside the ACB terminal. The heat exchanger is a coil through which chilled water or hot water circulates for cooling or heating purposes, respectively. A mixture of the primary air and induced air forms the supply air that is discharged to the occupied zone. A schematic diagram of a typical ACB terminal is shown in Figure 1.3.

Figure 1.2 Structure of ACB

Figure 1.3 Schematic diagram of ACB

ACB systems possess a lot of advantages compared with other conventional ACMV systems.

First, ACB systems can save energy in many ways.

- Typically, VAV systems meet the cooling requirement by adjusting the volume of cold air supplied. In ACB systems, the cooling coil covers most of the cooling duty, which decouples the primary airflow from cooling loads. The fan only needs to provide proper fresh air that meets the ventilation requirement. Thus, the ventilation fan energy consumption is reduced. In addition, water has a volumetric heat capacity 3500 times that of air, which can transfer much more cooling load than air. The fan energy is reduced by a factor of seven in typical pump and fan arrangements [20].
- ACB systems adopt higher chilled water temperatures (14°C to 18°C) than those of conventional ACMV system (4°C to 8°C). Thus, the chiller system operates at a 15% to 20% higher efficiency than that for a conventional system [13].
- For large spaces with a variety of rooms such as offices or laboratories, each space has its own general exhaust requirement as well as an independent reheat device. ACB systems avoid this energy loss by dynamically and individually cooling each space through adjusting the chilled water flowrate. In this case, ACB systems eliminate reheat energy and minimize outside air conditioning.

Second, the indoor environment quality (IEQ) of ACB systems is greatly improved in three aspects.

- As there is no fan near the occupied space, the noise of ACB systems is neglected. ACB systems normally work with noise criterion (NC) under 20, which is lower than the NC of 35-40 for traditional FCU systems.
- The fresh air ventilation and cooling loads can be separately controlled by the primary air and chilled water, which reduces the risk of draught problems while maintaining the room temperature.
- The unique terminal design can provide good Coandă effect, which refers to the tendency of staying attached to the convex surface for a fluid jet. This effect will ensure a good thermal comfort by avoiding spraying the airflow to the human body directly.

Third, due to the deduction of ventilation air, the size of the air duct and AHU can be decreased, which leads to an overall saving in construction and operation costs.

1.3 Motivations and objectives

The chilled beam system was first developed in 1975 [21], it's still a preliminary ACMV solution compared with the conventional ACMV systems. The research on ACB systems has not been sufficiently studied. Although ACB systems have many advantages theoretically, some of them have not been proved experimentally. There are many issues in practical operation of ACB systems. For example, ACB systems are always designed with oversized cooling capacity. The beams have to operate with excessive primary airflow. Since the primary air already provides adequate space cooling, the water valve is closed most of the time [22]. As a result, ACB systems often work as an expensive diffuser and the thermal comfort in the occupied zone is deteriorated. Condensation is also a concern for cooling ACB systems. The cooling coils inside ACB will condensate when the airflow is of high humidity and the coil is at low temperature. Therefore, the humidity condition needs to be observed and dew point shall not rise above the surface temperature of the chilled beams. In order to take full advantages of the potential of ACB

systems, the full-scale experiment is indispensable, and many parameters shall be monitored.

In modern society, the thermal discomfort and excessive energy consumption are the two major concerns for ACMV systems. Nowadays, people spend most of their time in air-conditioned rooms. The poor air quality and inferior thermal condition incur plenty of uncomfortable sensations and even give rise to sicknesses such as SBS. The most common and serious problem is draught, which is mainly caused by the cooling air movement. Therefore, it's of great importance to evaluate the air distribution and thermal comfort for ACB systems. However, the previous research on ACB systems is mainly focused on cooling capacity, energy-saving effect and humidity control. The studies on air distribution and thermal comfort are still inadequate to provide a comprehensive perspective on ACB systems.

The primary objective of this thesis is to have an overview of ACB systems and make the most of the advantages. Specifically, the objectives can be divided into the following Sections:

- Analyze the characteristics and develop precise models of airflow pattern for ACB systems.
- Investigate the full-scale distribution of air temperature, velocity, and humidity in the occupied zone.
- Evaluate effects of various factors on thermal comfort for ACB systems.
- Develop simplified models for ACB systems experimentally and numerically.
- Optimize the performance regarding to thermal comfort and energy efficiency for ACB systems.

1.4 Major contributions

The major contributions of this thesis are summarized as follows:

- A three-dimensional computational fluid dynamics (CFD) model for ACB terminal is developed to reflect air flow pattern inside ACB and how it is discharged from ACB nozzles. Experiments in accordance with the CFD model of ACB will be performed to verify the model. 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. Eventually, a proper strategy to solve this problem is proposed and validated by CFD simulation.
- A criterion is proposed for air detachment from the ceiling and a method is developed to judge the valid space for self-similarity of air jet. After being discharged for the ACB terminal unit, the air jet will be attached to the ceiling according to Bernoulli's Law. Then, the air jet will drop to the occupied zone due to the friction of the ceiling, the air entrainment effect, and buoyance effect. The further distance the detachment point is away from the terminal, the less possibility the air will spray to the occupied zone. This attachment of air jet, which also refers to Coandă effect, and the detachment have significant effect on the thermal comfort. Self-similarity is a unique characteristic for turbulent air jet, which can be used to predict the air velocity. Therefore, Coandă effect and self-similarity are studied experimentally in this thesis.
- The influences of the configuration and strength of heat sources on thermal comfort are analyzed. The overall thermal comfort performance of the ACB system is evaluated through full-scale experiments in the occupied zone. The common indices such as air diffusion performance index (ADPI), predicted mean vote (PMV), draught rate (DR) and vertical air temperature difference (VATD) are employed and investigated for each operating condition.
- An optimization method based on genetic algorithm (GA) is proposed for ACB systems. Simplified energy models for the fan, pump, and chillers, heat transfer model for the coil in ACB terminal, and velocity model are developed and validated by experiments. The energy consumption and thermal comfort is optimized by adjusting the primary airflow rate and water flow rate.

1.5 Organization of the thesis

This thesis is organized as follows:

Chapter 1 presents an overview of ACMV systems. The typical structure, working principles, and advantages of ACB systems are introduced. Motivations, objectives and major contributions for this thesis are also elaborated in this chapter.

Chapter 2 gives a comprehensive review on the previous research of ACB systems, including the effects of mechanical designs on the performance, model development on induction ratio and cooling capacity, characteristics of air jet for ACB terminals, methods to capture air velocity, factors that will affect airflow patterns and thermal comfort, criteria to evaluate thermal comfort performance, CFD applications and standards to specify the air distribution and thermal comfort.

Chapter 3 simulates the airflow condition for an ACB terminal unit with a three-dimensional CFD model. The un-uniformity air distribution is found by experiments. A proper strategy is proposed and verified by CFD simulation.

Chapter 4 investigates the velocity profiles of the air jet discharged by ACB terminals including the streamwise as well as the vertical direction experimentally. The effects of primary air pressure and supply air temperature on the airflow pattern are investigated. The characteristics of Coandă effect, self-similarity, and turbulence intensity for air jet are discussed.

Chapter 5 primarily evaluates the effects of configuration and strength of heat source on thermal comfort for ACB systems. Full-scale air temperature, speed and absolute humidity distributions are demonstrated for ACB systems. Based on the data, the thermal comfort is comprehensively evaluated with indices of ADPI, PMV, DR, and VATD.

Based on the obtained characteristics of air distribution and thermal comfort from Chapters 3 to 5, Chapter 6 proposes a GA optimization method for ACB systems. Simplified energy models, heat transfer model, and velocity model are developed and validated by experiments. The energy consumption and thermal comfort are optimized by adjusting the primary airflow rate and water flow rate.

Chapter 7 concludes the content of this thesis and explores the potential future work.

Chapter 2 Literature Review

2.1 Introduction

To achieve the objectives, a comprehensive literature review on ACB systems is necessary to be conducted. A good ACB terminal is the one that can provide satisfied cooling load on coil side with minimum primary airflow rate [22]. Therefore, the different kinds of mechanical designs of ACB terminals are essential to be discussed. In addition, the models proposed by other researchers are addressed in this chapter in order to accurately justify the performance of ACB systems. Indoor air quality, thermal comfort, contaminant distribution, and occupant productivities depend significantly on indoor airflow characteristics and indoor airflow distribution from supply air diffusers [23-26]. Thus, the previous research on air distribution and thermal comfort are thoroughly reviewed in this chapter. Due to the complex testing environment, some experiments are unable to be conducted and some data are not easily to be measured, a powerful software CFD is employed to simulate the airflow characteristics under various conditions. The related research is covered in this chapter.

2.2 Mechanical design of ACB systems

One significant feature of ACB system is the entrainment effect, which refers to the induction of room air caused by the negative pressure near nozzles. The index to evaluate this effect is called induction ratio (IR), which is defined as the ratio of induced airflow rate divided by primary airflow rate. IR is an important index for ACB system. Higher IR will lead to more heat transfer. As a result, more energy can be saved. Guan et al. [27, 28] investigated the geometry effect of ACB terminal on IR. It was concluded that the radius of nozzles was negatively correlated to IR, while the distance between nozzles was positively correlated to IR. IR can be increased by 30% by changing the geometry of the mixing chamber and lengthening the nozzle.

The circuitry arrangement of heat exchanger also has significant effect on the ACB performance. Can et al. [18] compared 1-circuit, 2-circuits, 4-circuits, and 8-circuits heat exchanger. It was found out that 2-circuits arrangement had the best performance instead of the 1-circuit heat exchanger that is widely used in ACB products. The heat exchanger of ACB terminal was further studied with respect to tube connecting sequences by Can et al. [29]. An optimized scheme for the heat transfer capacity was proposed with a particle swarm optimization program.

2.3 Models for ACB systems

Typically, the modellings on ACB systems are focused on two categories: IR and cooling capacity. As discussed in Section 2.2, the IR is an important factor for ACB systems. However, as the induced airflow rate is difficult to measure, many methods have been developed to calculate the IR index. Can et al. [30] proposed an empirical model for IR based on the pressure drop for primary airflow rate. The unknown coefficients were identified by experiments. However, the buoyancy effect was not considered in this model. ASHRAE standard 200 calculate IR through a function of primary air temperature, supply air temperature, and induced air temperature [31]. This method fails to precisely measure the induced air temperature. Because induced air temperature does not distribute uniformly. Peter et al. [32] proposed three methods to measure the induction ratio of ACB: velocity method, tracer gas method, and modified capacity method. They found that the tracer gas method was less consistent compared to other two methods. The modified capacity method avoided the positioning problem, but still overestimated 24% compared to the velocity method. Besides, the results indicated a higher chilled water temperature results in higher IR, while the room air temperature had no proportional relationship with IR. Guan et al. [28] built a three-dimensional CFD model as close as to the real experiment set. This CFD model agreed well with the experimental data with a deviation of $\pm 5\%$. Gammarata et al. [33] also built a numerical model in COMSOL Multiphysics. The IR values showed a very low sensitivity to the primary airflow rate.

As mentioned in Section 1.3, ACMV systems are normally designed with oversized cooling capacity to leave safety margins. In absence of individual room control, accurate cooling capacity models for ACB system are of great importance. The cooling capacity of ACB systems consists of two parts: cooling from primary air and cooling from chilled water in the coil. Standard of EN 15116 [34] specifies that the cooling capacity of coil side is a power function of the temperature difference between room air and chilled water. While, ASHRAE Standard 200 [31] claims that the cooling capacity of coil side can be represented by a heat transfer model. The coil heat transfer coefficient is governed by an empirical function of the mass velocity of induced air and the velocity of water in the coil. By utilizing the Modelica Standard Library, Maccarini et al. [35] proposed an alternative empirical model for heat transfer coefficient, which was a function of primary air mass flow rate per active length, temperature difference of water outlet and inlet, and water mass flow rate. Chen et al. [30] built an empirical hybrid model for forced convection heat transfer coefficient with respect to Reynolds number and Prandtl number. De Clercq et al. [36] developed models for heating and cooling capacities with number of transfer units (NTU) method.

2.4 Air distribution for ACB systems

Air distribution refers to how air is introduced to, flows through, and is removed from spaces [37]. The role of air distribution is to bring fresh and conditioned air to the occupied zone. Mixing ventilation (MV) and displacement ventilation (DV) are the most general mechanical ventilation applied in ACMV systems [38]. The airflow patterns of MV and DV are shown in Figure 2.1. MV systems provide air in a way that supply air and room air are fully mixed, which leads to small temperature variation and uniform contaminant concentration in the room. The supply air is normally of high velocity. In contrast, the DV systems supply the air directly to the occupied zone at low velocities to reduce the induction effect. The outlets are usually located near the floor level. The system utilizes buoyancy forces in the room, generated by heat sources such as people, lighting, computers, electrical equipment, etc., to remove heat and contaminants from the

occupied space. Due to the unique structure of ACB terminal units, they are normally installed near ceiling levels. Therefore, ACB systems mainly adopt MV as ventilation method.

Figure 2.1 Typical air distribution: (a) MV; (b) DV

One distinguished advantage of ACB system is that the air jet is attached to the ceiling after being discharged from the outlet because of the Coandă effect, which refers to the tendency of staying attached to the convex surface for a fluid jet. The Coandă effect for ACB system is illustrated in Figure 2.2. This effect has become increasingly used in a wide variety of applications, including medical, industrial, aerodynamics, and maritime technology [39]. It is shown that this effect can increase the air throw by up to 40% [40]. Making good use of the effect can ensure a good thermal comfort by avoiding spraying the airflow to the human body directly. Neuendorf and Wagnanski [41] observed from a circular cylinder that the adverse pressure gradient caused the detachment of the jet. Wu et al. [42] proved through experiments that ACB systems could generate good Coandă effect. The supply air from the ACB terminal was attached to the ceiling and risk of draught was greatly mitigated. The study of the detachment point is of paramount importance while analyzing airflow patterns. However, this effect is seldom discussed in previous research related to air distribution of ACB system.

Figure 2.2 Coandă effect

As the turbulent air jet has a unique characteristic of self-similarity, which refers to the velocities at different horizontal distances have similar profiles, as is presented in Figure 2.3. It can be elaborated as the ratio of local velocity (u) divided by the maximum velocity (U_m) has certain relationship with the local vertical position (y) divided by the vertical position when the velocity is equal to half of the maximum velocity ($y_{1/2}$).

Figure 2.3 Self-similarity

The self-similarity can be described by numerical models. Verhoff [43] proposed an empirical model:

$$u / U_m = 1,48\eta^{1/7} [1 - \operatorname{erf}(0.68\eta)] \quad (2.1)$$

where $\eta = y / y_{1/2}$ and $y_{1/2} = 0.068(x + 10s)$. s is the slot height, which is 0.04m for this ACB terminal unit. x is the horizontal distance from the outlet of ACB.

Schwarz and Cosart's model [44] is expressed as :

$$u / U_m = \exp[-0.937(\eta - 0.14)^2] \quad (2.2)$$

These two models are particularly used to describe the plane wall jet, which fit the characteristics of air jet discharged from ACB, in terms of the normalized velocity and normalized distance to the ceiling. These two models are widely used and compared in a lot of papers [45-47]. Awbi [48] compared these two models with the experimental results from Rajaratnam. He found that Verhoff's model gave better agreement. Cao et al. [49] utilized these two models to compare with the air jet vertically discharged under isothermal conditions. A good agreement was observed with Schwarz and Cosart's turbulence profiles at a distance downstream of more than 4 slot heights. It can be concluded that these two models have higher accuracy with simple expressions. However, this self-similarity is only valid under certain downstream distance after developed and before impinging on the wall. If the air jet is detached from the ceiling, the self-similarity is not suitable any more. A good similarity was revealed by Cao et al. [47] in the area at a distance of 600mm-2000mm in the case of $Re=2667$ and the data deviated from the theoretical profiles after 2000mm. They also found that the jet similarity may have relationships with the intensity of the turbulence and the initial Reynolds number of the jet [46].

The airflow pattern in the occupied zone is affected by various factors and has been studied by several researchers on these aspects. Zbořil et al. [50] evaluated the effects of heat sources such as occupants, computers and windows on the airflow pattern. They concluded that the thermal flows generated by the heat sources affected the airflow pattern significantly. Koskela et al. [51, 52] characterized the airflow pattern in an office

environment that was ventilated by ACB under summer, spring, autumn and winter cases. They proposed that the direction of the circulation was related to the warm and cool window in different seasons. Kosonen et al. [53] 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 with the rising of heat load. However, the maximum velocity was not significantly affected by the location of the thermal load. The un-uniform air distribution was easily led to turbulence, which had a high risk to induce draught sensation.

Because of its instability, several methods were developed to observe and analyze the air movement. Fredriksson et al. [54] examined the velocities and temperatures around the chilled beams based on a thermistor anemometer system. A laser-light visualization method was utilized to observe the dynamic process of the airflow discharged from chilled beams. Through experiment, it was proved that the airflow generated by a chilled beam exhibits strong fluctuations which may result in discomfort draught. The airflow pattern was hard to predict because it was extremely unstable and sensitive to the presence of heat sources. Cao et al. [49, 55] utilized an original method, particle image velocimetry (PIV), to trace airflow path with an approximate error of 20% at the low Reynolds numbers between 960 and 1680. The attached plane jet of an ACB was thoroughly studied and the attached plane jet moved in a turbulent way regardless of the Reynolds number. Besides, the flow oscillations had a notable effect on the growth rates of the jets when Reynolds numbers was between 960 and 2000. The flow field was also investigated by means of experiment and mathematical modelling based on isothermal and non-isothermal conditions. Specially, the air distribution in the corner region was thoroughly studied. The measurement results indicated that the existing jet model was unable to reflect the velocity decay due to corner effect. The conventional non-isothermal jet equation could not precisely predict the attached plan jet produced by ACB after turning at the corner.

Several standards are proposed to specify the methods for the measurement and evaluation of air conditions. ASHRAE Standard 62.1 can be referred to measure the

indoor air quality and used as a guide for various ventilation system [56]. The method of testing for room air diffusion based on air temperature distribution and air velocity is specified in ASHRAE Standard 113 [57].

2.5 Thermal comfort for ACB systems

The recent interests in the field of thermal comfort follow an exponential trend with a considerable increase in publications in the past decade [58]. Typically, thermal comfort is a subject perception for the occupants exposed to the air-conditioned environment. It varies from person to person physiologically and psychologically. Thus, it can hardly be judged. Fanger proposed an equation to numerically evaluate the thermal sensation in 1970 [59]. The main factors that affect thermal comfort are what determine heat gain and loss, namely clothing insulation, metabolic rate, mean radiant temperature, air temperature, relative humidity and air speed [60].

Draught is a kind of uncomfortable sensation of air movement, which is the major reason to damage thermal comfort. Dedicated experiments were carried out to explore this draught problem. True et al. [61] pointed out different designs related to the location of the chilled beams, the number, size and location of heat sources, the dimension of the room and the primary and induced airflow rates may cause draught discomfort through several experimental comparisons. It was concluded that the height of the rooms equipping with chilled beams should no more than 3.5 m and high velocity level may lead to unnecessary draught risks. Kosonen et al. [62, 63] argued that the key factors to estimate the risk of a draught were temperatures of the room air and air jets, local air velocity, and fluctuations. Convection flows were studied by experiments in a full-scale room ventilated by ACB. The results showed that convection flow had less impact when the chilled beam was installed in the longitudinal direction than when it is installed in the latitudinal direction. They also compared the radiant temperature and radiant temperature asymmetry between chilled beam systems and radiant panel systems. It was found that the difference between them was very small. Furthermore, the difference between the

operative temperature and the air temperature was not significant.

As the thermal comfort is a subjective feeling to the environment, it is difficult to judge its performance. Therefore, several criteria, such as air diffusion performance index (ADPI), predicted mean vote (PMV), draught rate (DR) and vertical air temperature difference (VATD), are proposed to numerically evaluate the air diffusion and thermal comfort. ASHRAE Standard 55 addresses factors primarily influencing the comfort perception of people [64]. ISO 7730 presents two main indices: PMV and PPD, as the criteria to predict the thermal sensation and degree of discomfort [65].

ADPI is one of the important indices which are utilized to evaluate the rating of air diffusion performance in the room. Based on the ASHRAE standards 113 [57], most people will feel comfortable under sedentary conditions where the effective draught temperature T_{ed} is between -1.7°C and $+1.1^{\circ}\text{C}$ and the air speed is less than or equal to 0.35m/s . ADPI is the percentage of test points that meet the criteria. Three factors of local air temperature, mean air temperature and air speed are considered. The humidity condition is not incorporated in this index.

$$\text{ADPI} = \frac{\text{Number of test points that meet criteria \%}}{\text{Total number of test points}} \quad (2.3)$$

The criteria are described as:

$$-1.7^{\circ}\text{C} < T_{ed} < +1.1^{\circ}\text{C} \text{ and } v < 0.35\text{m/s} \quad (2.4)$$

$$T_{ed} = (T - T_{avg}) - 8.0 \times (v - 0.15) \quad (2.5)$$

PMV, proposed by Fanger [59], is an comprehensive index based on votes from a large group of people. It scales the thermal sensation into 7 levels (Table 2.1). The thermal discomfort is represented PPD, which is an expression of PMV. It can be obtained according to the following equations based on standards of ISO 7730 [65]:

Table 2.1 Seven-point thermal sensation scale

+3	Hot
+2	Warm
+1	Slightly warm
0	Neutral
-1	Slightly cool
-2	Cool
-3	Cold

$$PMV = [0.303 \cdot \exp(-0.036 \cdot M) + 0.028] \cdot \left\{ \begin{aligned} &(M - W) - 3.05 \cdot 10^{-3} \cdot [5733 - 6.99 \cdot (M - W) - p_a] - 0.42[(M - W) - 58.15] \\ &- 1.7 \cdot 10^{-5} \cdot M \cdot (5867 - p_a) - 0.0014 \cdot M \cdot (34 - t_a) \\ &- 3.96 \cdot 10^{-8} \cdot f_{cl} \cdot [(t_{cl} + 273)^4 - (\bar{t}_r + 273)^4] - f_{cl} \cdot h_c \cdot (t_{cl} - t_a) \end{aligned} \right\} \quad (2.6)$$

$$t_{cl} = 35.7 - 0.028 \cdot (M - W) - I_{cl} \cdot \left\{ 3.96 \cdot 10^{-8} \cdot f_{cl} \cdot [(t_{cl} + 273)^4 - (\bar{t}_r + 273)^4] + f_{cl} \cdot h_c \cdot (t_{cl} - t_a) \right\} \quad (2.7)$$

$$h_c = \begin{cases} 2.38 \cdot |t_{cl} - t_a|^{0.25} & \text{for } 2.38 \cdot |t_{cl} - t_a|^{0.25} > 12.1 \cdot \sqrt{v_{ar}} \\ 12.1 \cdot \sqrt{v_{ar}} & \text{for } 2.38 \cdot |t_{cl} - t_a|^{0.25} < 12.1 \cdot \sqrt{v_{ar}} \end{cases} \quad (2.8)$$

$$f_{cl} = \begin{cases} 1.00 + 1.290 I_{cl} & , \text{ for } I_{cl} \leq 0.078 \text{ m}^2 \cdot \text{K} / \text{W} \\ 1.05 + 0.645 I_{cl} & , \text{ for } I_{cl} > 0.078 \text{ m}^2 \cdot \text{K} / \text{W} \end{cases} \quad (2.9)$$

$$PPD = 100 - 95 \cdot \exp(-0.003353 \cdot PMV^4 - 0.2179 \cdot PMV^2) \quad (2.10)$$

where M is the metabolic rate; W is effective mechanical power; I_{cl} is the clothing insulation; f_{cl} is the clothing surface area factor; t_a is the air temperature; $\bar{t}_r$ is the mean radiant temperature; v_{ar} is the relative air velocity; p_a is the water vapor partial pressure; h_c is the convective heat transfer coefficient; t_{cl} is the clothing surface temperature.

The DR is used to predict the percentage of occupants who are suffering from the unpleased sensation of the body caused by the air movement. It is normally calculated from the following model. This model usually applies to sedentary activity at neck level (1.1m). It could overestimate the draught rate at arms and feet level. The maximum DR is adopted to represent the draught condition. It can be calculated as follows [65]:

$$DR = (34 - t)(\bar{v} - 0.05)^{0.62} (0.37 \cdot \bar{v} \cdot TI + 3.14) \quad (2.11)$$

The turbulence intensity is calculated as [47]:

$$TI = \frac{S_v}{\bar{v}} \cdot 100\% \quad (2.12)$$

where $\bar{v}$ is the average velocity; S_v is the standard deviation for air jet velocity.

The thermal comfort can be divided into three categories (Table 2.2) based on ISO 7730.

Table 2.2 Categories of thermal environment

Category	PPD (%)	PMV	DR (%)	VATD (°C)
A	<6	-0.2<PMV<0.2	<10	<2.7
B	<10	-0.5<PMV<0.5	<20	<3.3
C	<15	-0.7<PMV<0.7	<30	<4.2

The uniformities for air speed, air temperature and absolute humidity are evaluated through the standard deviation, which is defined as

$$S_v = \sqrt{\frac{1}{N-1} \sum_{i=1}^N (x_i - \bar{x})^2} \quad (2.13)$$

Rhee et al. [66] compared the performance of the conventional air distribution system, ACB system and underfloor air distribution system (UFAD) with respect to thermal uniformity. ADPI, air velocity and vertical air temperature difference were obtained through comparative experiments. ACB system achieved a good thermal performance with 80.7% percent ADPI, category B in accordance with ISO 7730 and a low VATD which is less than 1°C. The thermal comfort were compared through the methods of comfort enquiry and experimental measurements [67]. The number of dissatisfied votes

were mapped and contrasted to the standard PMV/PPD curve. The author thought that the measured mean PMV didn't revealed any relationship based on responses of the subjects. The standards may help to evaluate the thermal condition however it should not be relied on as absolute references. Wu et al. [68] evaluated the effects of heat sources on thermal comfort for ACB systems by means of ADPI, PMV, DR, and VATD. The results illustrated that ADPI, PMV, and DR were significantly affected by air speed. The large air speed appeared mostly at the ankle level (height 0.1m). Thus, the ankle level had a higher risk to feel uncomfortable.

Previous research have found that the thermal comfort is significantly affected by various factors such as the heat sources, types and configurations of the ACMV terminals, location of the inlet and outlet, human activities and outdoor environment [58, 61, 69-72]. The heat sources in office environment, which are normally comprised of PCs, lightings, occupants, solar radiation and warm window, can significantly change the air temperature, air velocity, and humidity in the room. Thus, the heat sources play an important role in affecting the thermal condition. Melikov et al. [73, 74] explored the interaction of flows in a room ventilated by four active chilled beams. He found that the increase of heat load and primary airflow rate reinforced the non-uniformity of the thermal environment in the room can increased dissatisfied percent of subjects due to draught.

Studies also show that the turbulence intensity of the air jet is closely relevant to thermal performance. It was proved that a turbulent air brought more uncomfortable feeling than a laminar air regardless the velocity value [75-77]. Mayer [78] suggested that the heat transfer coefficient would increase when the standard deviation of the fluctuating velocity was increased. Melikov et al. [79] found that the turbulence intensity was higher at low mean speeds and gradually decreased with the mean speed until an asymptotic value was attained. Fanger et al. [80] studied the influences of turbulence intensity on the sensation of draught. The results showed that the feeling of draught was significantly influenced by the turbulence intensity. They also developed an empirical model for percentage of dissatisfaction as a result of draught in the head region taking into consideration air temperature, mean air speed and turbulence intensity.

2.6 CFD applications

CFD software has been widely utilized in field of fluid analysis as an efficient tool to predict the air movement since 1970s [81-85]. Besides, it also plays an important part in visualizing the flows in three-dimensional spaces [45, 86, 87]. The airflow in the room follows the law of energy conservation, which is described as the Navier-Stokes equations. The procedure of solving these equations through a numerical method is called computational fluid dynamics [88]. With the rapid development of ACMV systems, CFD has made a great contribution in analyzing the fluid and thermal dynamics.

Cammarata et al. [33] evaluated the fluid-dynamical and thermal performances of chilled beam system based on COMSOL Multiphysics. Two main parameters were analyzed: the induction rate and the thermal capacity. The velocity fields computed showed an almost constant distribution in the portion of the considered standard room supposed occupied by human presence. Varying the imposed inlet velocity of air flow through the chilled beam nozzles, velocity vectors quantitatively changed closed to the inlet components only. However, there was no authentic experiment to validate this model. Hannu et al. [52] also simulated the distribution of air velocity and temperature of an open-plan and a single-person office in three kinds of climate-summer, winter and midseason (spring/autumn) based on CFD. It turned out that the mean air speed of the simulation was close to the real result. However, the maximum velocities were smaller than actual measurement. Einberg et al. [89] predicted the velocity and temperature 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 flow from a displacement diffuser and jet flow. Cehlin and Moshfegh compared four turbulence models in computational fluid dynamics, i.e. the standard $k-\epsilon$, the LVEL, the Chen-Kim model and the RNG models regarding the temperature and air flow performances near a low-velocity diffuser. It turned out that the Standard $k-\epsilon$ model and RNG model were much more stable than the Chen-Kim model [90]. A numerical method named Micro/Macro Level Approach (MMLA) was developed by Almansa et al. [86] to model a conventional

diffuser with benefits of reducing the working load of points to solve the airflow in a room. A new visualization methodology called the oil droplet was adopted in the experiments which had higher operability than PIV and hot-wire anemometry. Furthermore, CFD models for a diffuser and for the room were built. The result revealed that the experimental data showed a good agreement in comparison with the CFD result. A simplified CFD model was developed by Hannu for nozzle duct diffuser due to complicated modelling of flow pattern near the nozzle duct in practical CFD simulations. The performance of the model was validated against laboratory measurements [91]. Wu et al. [92] analyzed the airflow patterns of an ACB terminal with a three-dimensional CFD model. An un-uniform air distribution was found near the nozzles. A proper strategy was proposed and verified through simulation.

Overall, CFD is a useful tool to help evaluate air distribution especially when the experiments are difficult to be conducted. Substantial energy and time will be saved by utilizing CFD simulation. Nevertheless, the researches on CFD simulation for ACB systems are not sufficient.

2.7 Summary

This chapter demonstrated a comprehensive review of research on ACB systems. The research mainly focuses on the mechanical design of ACB terminals, models for ACB systems, air distribution, thermal comfort and CFD applications. The air distribution will interact with thermal comfort to a large extent. The varying of air distribution will change the comfort zone while the thermal comfort factors like heat sources, would also influence the distribution of air flow. The research for air distribution and thermal comfort will have an enormous influence on the improvement of indoor air environment.

Chapter 3 CFD simulation for airflow patterns

3.1 Introduction

As discussed in Section 2.4, the air distribution has a significant impact on thermal comfort. The un-uniformity of air distribution will easily give rise to turbulent flow which may cause displeased feeling such as draught in the occupied space. ACB terminal 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. Although there are several studies on air distribution in the occupied place, few mention the airflow inside the ACB. The non-uniformity of the airflow has a huge potential to cause uncomfortable draught for occupants. Air distribution for a two-way discharge ACB terminal is investigated in this chapter. A three-dimensional CFD model for ACB terminal is built in Section 3.2. The model is validated by experiments in Section 3.3. The air velocities around the nozzles under different conditions are tested simulated by the three-dimensional CFD model. The un-uniformity air distribution is found by experiments. A proper strategy is proposed and verified by CFD simulation in Section 3.4. A brief summary for this chapter is presented in Section 3.5.

3.2 CFD modeling of ACB

3.2.1 Computational grid

A three-dimensional ACB model is developed by computational fluid dynamics (CFD). As the ACB terminal is symmetry from left to right side, only half of the terminal was built to simplify the calculation. The computational grid is formed by tetrahedron cells (Figure 3.1) because of the complex structure of the ACB terminal. The minimum element quality of this CFD model is 0.213 and the cell number is 2595478. In order to

verify the mesh independence, another simulation is performed with 3739208 cells for the case of nozzle diameter is 21.5mm and primary pressure is 30 Pa. Assume the inlet side is the back side of ACB, where nozzles are sequenced from 31 to 45. On the contrary, points 1-15 are nozzles located in the opposite side of inlet. The velocities of nozzles 1-15 are compared in Figure 3.2. As the maximum velocity difference is 0.039 m/s, which is acceptable, the mesh independence is validated.

Figure 3.1 CFD model mesh

Figure 3.2 Velocity comparison for two mesh models for nozzles 1-15

3.2.2 Boundary conditions

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 primary air pressure is set to be 30Pa. The outlet pressure and induced air inlet pressure is 0 Pa. Other parameters are all default values.

3.2.3 Solver settings

The standard k- ϵ model, which has been proved to be suitable for ACB system in [28], is adopted as the turbulence model. The SIMPLEC algorithm is used for pressure-velocity coupling. Second order upwind discretization scheme is applied in the solving momentum, turbulence energy and turbulent dissipation rate terms.

3.3 CFD model validation

This CFD model is verified by experiments. An ACB terminal (Figure 3.3) with a dimension of 1200mm*600mm*200mm 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 of this velocity transducer is 10 m/s with an accuracy of $\pm 0.5\%$. 120 data are collected for each nozzle under steady state, when the primary air pressure of ACB is within 3Pa for 60 minutes. The experimental results with error bars are shown in Figure 3.4.

The nozzle velocity comparison between simulation and experiment is shown in Figure 3.5. The velocities match well with the simulation velocities except for a few middle points. The relative error between experiment and simulation are also calculated (Figure 3.6). Most relative errors are within 20%, which can be acceptable for CFD simulation.

Figure 3.3 The ACB terminal

Figure 3.4 Experimental results with error bars

(a)

(b)

Figure 3.5 Nozzle velocity comparison between simulation and experiment: (a) front nozzles; (b) back nozzles

Figure 3.6 Relative error between experiment and simulation

3.4 Results and analysis

3.4.1 Simulation results

The simulation air streamlines are shown in Figure 3.7, from which it can be seen that the air velocity distribution is quite fluctuant in the primary air plenum. It can be drawn from the line charts (Figure 3.8) 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 side areas. 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.

In order to investigate the effects of nozzle diameter and primary air pressure on this velocity un-uniformity between front and back nozzles, the CFD model is applied to simulate the velocities for nozzle diameter 21.5 mm, 15 mm and 9 mm under primary air pressure of 30 Pa, 60 Pa and 90 Pa, respectively. The simulation results are depicted in Figure 3.9 to Figure 3.11.

Figure 3.7 ACB simulation streamlines for nozzle diameter 21.5mm under 30 Pa

Figure 3.8 Simulation velocities for nozzle diameter 21.5mm under 30 Pa

Figure 3.9 Velocity differences for nozzle diameter 21.5mm under 30 Pa, 60 Pa, 90 Pa

Figure 3.10 Velocity differences for nozzle diameter 15mm under 30 Pa, 60 Pa, 90 Pa

Figure 3.11 Velocity differences for nozzle diameter 9mm under 30 Pa, 60 Pa, 90 Pa

As shown in Figure 3.9 to Figure 3.11, the velocity difference reduces with a decrease of nozzle diameter. When the nozzle is small enough, the velocity differences can be neglected. Moreover, for the same nozzle diameter, the higher the primary air pressure, the bigger the velocity gap between two lines nozzles. The maximum velocity differences for each case are depicted in Table 3.1.

Table 3.1 Maximum velocity differences

Pressure Nozzle 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

3.4.2 An optimized strategy to improve un-uniformity of nozzle velocity

In order to solve this unpleasant un-uniformity, several solutions have been proposed and tested. One of the strategies is to add a board (200*70*2mm) in the middle of the primary air plenum as shown in Figure 3.12.

Figure 3.12 Modified ACB terminal

The velocity difference is defined as :

$$\Delta v = v_{f,i} - v_{b,i} \quad (3.1)$$

where v_f is the velocity of the front nozzle; v_b is the velocity of the back nozzle; i is the sequence of the nozzles.

As is depicted in Figure 3.13, the velocities differences between front 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 that of 0.957 m/s for the original ACB. It can be concluded that adding a board in the middle will significantly improve the uniformity of velocity distribution.

Figure 3.13 Velocity differences between front and back nozzles

3.5 Summary

From the results of simulation and experiments of air distribution for ACB nozzles, the conclusion could be drawn that the un-uniformity was significant when the nozzle diameter was large. Moreover, the rise of the primary air pressure would aggravate this un-uniformity. The smallest velocities were located close to inlet of back side, while the largest velocities were distributed in the nozzles of front side and the two verges. The pressure around the opposite side of inlet and edge area was comparatively larger which may leads to the high speed in those areas. However, this troublesome un-uniformity would be eliminated when the nozzle was small enough. When nozzles were small, the air flow velocity was primarily restricted by the pore size. However, for the nozzles owning a large pore size, the air velocity would 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 primary air pressure are large.

In addition, a solution of adding a board in the middle was proposed and validated by CFD simulation. With the help of this board, the air would be discharged evenly from the nozzles. Through a complex process, the air would be distributed to all of the nozzles in a more balanced way. 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 few chances of draught problem.

Chapter 4 Experimental investigation on airflow patterns

4.1 Introduction

It is found from previous chapter that the air distribution is not uniform at the nozzles of ACB terminals. A further experimental investigation on the air distribution in the occupied zone is of great importance to ensure a satisfied thermal comfort environment. The research concerning airflow pattern of ACB systems in the occupied zone has not been sufficiently studied. Some of the research is conducted within a limited testing area or under the condition that the ACB terminal is placed on the ground [46, 47, 55]. The velocity profiles along streamwise direction, which are meaningful in analyzing the attachment of air jet to the ceiling, have not been fully presented under operating conditions. Currently, most of the evaluations on the airflow pattern, for example, heat sources, outer environment and arrangement of ACB terminal unit, are external factors. The influences of internal factors, such as the airflow rate and supply air temperature, are seldom taken into consideration.

To cover these gaps and achieve a comprehensive understanding of the airflow pattern of ACB system, the velocity profiles of the air jet discharged by the ACB terminal are investigated in this chapter. The experimental setup, including the ACB terminal unit, the test room, and measurement devices, is illustrated in Section 4.2. The velocity contours and profiles for airflow from the ACB terminal are demonstrated in Section 4.3. The Coandă effect, self-similarity, and turbulence intensity are discussed in Section 4.4. A brief summary for this chapter is presented in Section 4.5.

4.2 Experimental setup

4.2.1 The ACB terminal

The ACB used in experiment (Figure 4.1 (a)) is a two-way terminal unit with a dimension of 1200mm (length)*600mm (width)*200mm (height). 29 circular rubber nozzles with diameter 9mm are evenly distributed on each side at the interface of primary air plenum and mixing chamber. The diameter of the primary air inlet is 150mm and the size of the outlet is 1200mm (length) *70mm (width).

Figure 4.1 (a) ACB terminal unit; (b) Thermal isolated room

4.2.2 The test room and measurement point distribution

The experiments are conducted in a 7.3m*3.3m*2.5m (L×W×H) thermal isolated room (Figure 4.1 (b)). As ACB is installed in the middle of the room, the airflow pattern can be considered symmetry on the two sides. Thus, the air velocity is tested on one side covering nearly half of the room. The regions near the wall are ignored because the air velocity is highly influenced by the wall. As the air jet is attached to the ceiling after being ejected for the outlet, testing area is set at the vicinity of the ceiling as a vertical plane in the middle of the outlet with dimension of 2700*300mm (Figure 4.2). 60 (6*10) points are measured with a horizontal interval of 300mm and vertical interval 50mm. The jet slot height for this ACB terminal unit is 40mm, which refers to the perpendicular distance for pathway of the supply air (as shown in Figure 4.2). This definition is referred to the work of Cao et al.[46]'s.

Figure 4.2 Measurement point distribution

4.2.3 Measurement devices

The air velocity is measured by an omnidirectional hot wire air velocity transducer, whose full scale is 2.5m/s. It uses a probe to measure the air velocity, which can capture the airflow from all directions. The transducer is equipped on a mobile device, which can take the velocity transducer to the corresponding position with an error of $\pm 5\%$. The velocity at the outlet of ACB is tested by a different type of air velocity transducer with a range from 0-10m/s. The primary air pressure of ACB is tested by a differential pressure transmitter. One side is connected to the atmosphere and the other side is linked to the inlet of ACB terminal unit. The supply air temperature is measured by a PT1000 temperature transducer. The primary airflow rate is tested by a KIMO multifunction transmitter with DEBIMO blades in the primary air duct and the accuracy is 5%, which meets the requirement of ASHRAE standards 200. The induced airflow rate is measured by the TSI air capture hood together with a compensation fan based on the zero-pressure method according to ASHRAE Standard 200. The sensors' details are shown in Table 4.1.

Table 4.1 Sensor specifications

Sensor	Application	Type	Range	Accuracy	Standard
Air velocity transducer	Room air velocity	TSI 8475	0-2.5m/s	$\pm 3\%$ reading, $\pm 1\%$ full scale	ASHRAE Standard 113
Air velocity transducer	Outlet air velocity	TSI 8455	0-10m/s	$\pm 2\%$ reading, $\pm 0.5\%$ full scale	ASHRAE Standard 70
Differential pressure transmitter	Primary air pressure of ACB	Dwyer MS-111	0-250Pa	$\pm 2\%$	ASHRAE Standard 200
Temperature transducer	Air temperature	S+S HFTM	0-50°C	$\pm 0.2^\circ\text{C}$	EN 15116
Multifunction transmitter	Primary airflow rate	KIMO C310	0-9216m ³ /h	$\pm 5\%$	ASHRAE Standards 200
Micromanometer	Induced airflow rate	TSI 8710	42 to 4250m ³ /h	$\pm 3\%$	ASHRAE Standards 200

4.2.4 Measurement procedures

The inlet static pressure of 50Pa-250Pa is recommended by the active and passive beam application design guidebook [93]. Three primary air pressures 50Pa, 100Pa and 150Pa, which correspond to three different supply airflow rates, are applied as variables for isothermal cases. As 24°C is the set temperature for this thermal isolated room, it is chosen as the supply air temperature for isothermal cases. According to the guidebook, chilled-water temperature is suggested between 14°C-18°C. To meet this requirement, deductions of 3 °C and 6 °C are imposed to the supply air temperature to investigate the non-isothermal conditions. Totally 5 cases are investigated under isothermal and non-isothermal conditions. The principle for designing the isothermal and non-isothermal experiments is to investigate the airflow pattern when the temperature of the primary air is different from the zone temperature. Case 1 is the basic case to be contrasted with other cases. The specific measurement condition is demonstrated in Table 4.2.

Due to wide testing points and the limited experiment condition, the transient air velocity distribution is hardly to be achieved simultaneously. The air velocity is tested one point by one point. However, the testing time delay has little effect on the result because the experiment is conducted under stable initial condition. The stable initial condition is

considered to be obtained when the standard deviation for primary air temperature is within 0.1°C and for primary air pressure of ACB is within 3Pa for 60 minutes based on EN standards 15116. As the air jet is turbulent air, the air velocity fluctuates continually within a certain range. As a result, the velocity is measured with a sample time of 120 seconds. The sample rate is set as 1 Hz.

The Reynolds number for supply air of ACB is calculated from the equation:

$$Re = vL / \nu = \rho vL / \mu \quad (4.1)$$

where v is the velocity of the supply air; L is the characteristic length which is the slot height in this chapter; ν is the kinematic viscosity of the supply air; μ is the dynamic viscosity of the supply air; ρ is the density of the supply air.

The Archimedes number is also calculated from the following equation to evaluate the buoyancy effects.

$$Ar = \beta g H \Delta T / u_0^2 \quad (4.2)$$

where β is the volume expansion coefficient; g is the gravitational acceleration; H is the room height; ΔT is the temperature difference between the supply air and the room air; u_0 is the initial velocity of the supply air.

Table 4.2 Measurement conditions

Cases	Primary air pressure	Supply air temperature	Room temperature	Differential temperature between supply air and room air	Reynolds number	Archimedes number	Maximum velocity standard deviation
1	50Pa	24°C	24°C	0°C	5156	0	0.101
2	100Pa	24°C	24°C	0°C	6456	0	0.075
3	150Pa	24°C	24°C	0°C	8522	0	0.105
4	50Pa	21°C	23°C	2°C	5261	0.0414	0.086
5	50Pa	18°C	22.7°C	4.7°C	5358	0.0973	0.067

4.3 Results

Totally 60 points are measured in the testing area. 120 velocity data are collected for each point. The average velocity is calculated as the velocity of the related point in order to reduce the error. Based on the data, the velocity contour, the velocity profiles along streamwise direction and the velocity profiles along vertical direction are plotted and analyzed in the following part.

4.3.1 Velocity contours

The velocity contours depicted in Figure 4.3 for all cases are plotted by Matlab. The horizontal axis stands for the horizontal distance from the ACB outlet, while the vertical axis represents the vertical distance from the ceiling. Case 1 is plotted repetitively with different velocity scale for the purpose of comparison. As for isothermal cases (Figure 4.3 (a)), it can be seen that the air jet expands as soon as it is discharged from the outlet of the slot as a consequence of the entrainment of the surrounding air. The spreading angle shown from the velocity contour is smaller when the primary air pressure is higher. In addition, the air jet stays attached to the ceiling over a long downstream distance for all three cases. The velocity decay is also clearly shown through contours. With respect to the non-isothermal cases (Figure 4.3(b)), an obvious air jet bending in the middle of the half-room can be observed when supply air temperature is at 18°C and the corresponding temperature difference between supply air and room air is 4.7°C. This phenomenon reflects that the air jet starts to detach from the ceiling after a distance around 1.2m for case 5. By comparing from these three cases, the bigger the temperature difference between supply air and room air, the more apparent this detachment is.

(a)

(b)

Figure 4.3 (a) Velocity contours for isothermal conditions (cases 1-3); (b) Velocity contours for non-isothermal conditions (cases 1, 4, 5)

4.3.2 Velocity profiles along streamwise direction

Velocity profiles along streamwise direction for all cases are depicted in Figure 4.4. 'y' denotes the vertical distance from the ceiling. For isothermal cases (Figure 4.4 (a)), it can be clearly seen that the velocities along streamwise direction at $y=0.05\text{m}$ for all cases descend sharply after dispersed from the outlet of the slot. The decrease of the velocity is mainly because of momentum transfer as a result of entraining surrounding air. The shear stress caused by the ceiling also has an impact on the velocity of air jet close to the ceiling. The velocity profiles along streamwise direction have similar paths at $y=0.15\text{m}$, 0.2m , 0.25m and 0.3m . With the increase of the horizontal distance from the slot, the air velocity will rise slowly first, then decrease and has a tendency to reach the same value at the end. As velocity rising is delayed and the magnitude of velocity is reduced for the airflow located at lower height (farther to the ceiling), it can be inferred that the air jet will entrain the surrounding air layer by layer and the airflow at lower height will be entrained later. When the airflow at different layers achieves the same velocity, the mutual entrainment will terminate. The peak velocities will shift to the right when the primary air pressure is higher. For example, the peak velocity along streamwise direction is situated at $x=1.51\text{m}$, $x=1.81\text{m}$ and 2.11m for $y=0.2\text{m}$ for case 1, case 2 and case 3, separately (Figure 4.4 (b)).

As for the non-isothermal conditions (Figure 4.4 (c)), the velocity profiles along streamwise direction have a drastic decrease after reaching to the maximum compared to the isothermal cases. When the temperature difference between supply air and room air is bigger, the peak velocity shifts to the middle and the air jet detaches earlier from the ceiling. The velocity starts to decrease at around 1.8m , 1.5m and 1.2m for case1, case 4 and case 5 separately (Figure 4.4 (d)), which corresponds with the air detachment positions in Figure 4.3. The streamwise peak velocity can be considered as a sign for the air jet to begin to detach from the ceiling. After comparing the isothermal and non-isothermal cases, it can be drawn that the large air temperature difference will accelerate the detachment, while the high primary air pressure will slow down this detachment.

Figure 4.4. (a) Velocity profiles along streamwise direction for isothermal conditions (cases 1-3); (b) Velocity profiles along streamwise direction for isothermal conditions (cases 1-3) at $y=0.15\text{m}$; (c) Velocity profiles along streamwise direction for non-isothermal conditions (cases 1, 4, 5); (d) Velocity profiles along streamwise direction for non-isothermal conditions (cases 1, 4, 5) at $y=0.15\text{m}$; where y is vertical distance from the ceiling

4.3.3 Velocity profiles along vertical direction

In order to study the self-similarity of the air jet, the velocity profiles along vertical direction are also investigated and demonstrated in Figure 4.5. To distinctly observe the vertical velocity profiles, the profiles at different horizontal distances are separated. The relative value between the solid line and the dashed line with same color denotes the velocity magnitude at that point. The vertical maximum velocity is crossed with a black dashed line, which can reflect the air jet trend to a certain degree. For isothermal cases shown in Figure 4.5(a), it is indicated from this black dashed line that the vertical maximum velocities are attached to the ceiling for a Section of distance after outlet. Then the peak velocity starts to detach from the ceiling. The vertical velocity has similar profile at $x=0.31\text{m}$, $x=0.61\text{m}$, $x=0.91\text{m}$ and $x=1.21\text{m}$ for case1. For the location at $x=0.01\text{m}$, it's too close to the outlet where the air jet is not well developed. When the peak velocity line is detached from the ceiling, the vertical velocity profile is gradually skewed. When $x=2.71\text{m}$, the airflow is highly effected by the wall, which will not be taken into consideration in this chapter. For non-isothermal cases (Figure 4.5 (b)), it can be seen that the vertical velocity profiles are highly influenced by the supply air temperature. The peak velocity line is detached from the ceiling earlier for lower supply air temperature.

(a)

(b)

Figure 4.5 (a) Velocity profiles along vertical direction for isothermal conditions (cases 1-3) as well as the change in vertical peak velocity; (b) Velocity profiles along vertical direction non-isothermal conditions (cases 1, 4, 5) as well as the change in vertical peak velocity

4.4 Discussion

4.4.1 Coandă effect

As shown in Figure 4.3(a), the air jet stays attached to the ceiling over a long downstream distance for all isothermal cases. It can be concluded that ACB system can provide good Coandă effect for isothermal condition under 50-150 primary air pressure. However, the differential temperature of supply air and room air will dramatically alter the airflow pattern, as a result, the Coandă effect is weakened. The air jet clings to the ceiling longer for higher primary air pressure as the higher primary air pressure leading to higher momentum ensures better attachment. In contrast, the differential temperature of supply air and room air causes the air to detach from the ceiling due to the air density difference, which may be harmful to the thermal comfort if it turns to the area which is the active zone for human beings. In other words, temperature difference between supply air and room air will damage the Coandă effect. Therefore, both primary air pressure and differential temperature of supply air and room air should be taken into consideration when the thermal comfort is highly required.

4.4.2 Self-similarity of the air jet

In order to test the self-similarity, the value of u , y and U_m can be achieved from the experiment. $y_{1/2}$ is calculated through interpolation method of "Piecewise Cubic Hermite Interpolating Polynomial" (PCHIP) based on Matlab [94]. The relationship of y and normalized velocity (u/U_m) is plotted in Figure 4.6(a) for isothermal cases and Figure 4.6(b) for non-isothermal cases. It can be found out that some curves are distorted at right bank and $y_{1/2}$ is unable to be obtained. After comparing with Figure 4.5, these twisted curves are the ones that vertical maximum velocity goes beyond $y=0.05\text{m}$, which is 1.25 slot heights. It can be inferred that when the vertical maximum velocity is beyond around 1.25 slot height, the self-similarity is not suitable for the air jet. This result may

explain why the data after 2000mm does not fit the theoretical profile in Cao et al.'s report [47].

Based on the conclusion above, the two models are also compared with the experimental data whose vertical maximum velocity is at 1.25 slot height ($y=0.05\text{m}$) for case 3 in Figure 4.7. It can be seen that the data fit the two models well. Mean absolute errors are calculated for the two models in Eq. (4.3). Mean absolute errors are calculated according to the following formula:

$$\text{MAE} = \frac{1}{N} \sum_{i=1}^N |e_i| \quad (4.3)$$

where MAE is the mean absolute error, N is the vertical testing points, which is 6 in this chapter, e_i is the error between the experiment result and calculated result. It can be drawn from Figure 4.8 that Schwarz and Cosart's model describes the velocity profile better at the location close to the outlet of ACB. However, Verhoff's model has a better performance when the air jet is a little bit far away from the outlet. Overall, Verhoff's model describes the self-similarity more precisely.

Figure 4.6 (a) Relationship of y and normalized velocity (u/U_m) for isothermal conditions (cases 1-3); (b) Relationship of y and normalized velocity (u/U_m) for non-isothermal conditions (cases 1, 4, 5); where x is the horizontal distance from the outlet of ACB

Figure 4.7 Model comparison

Figure 4.8 Mean absolute errors of two models

4.4.3 Turbulence intensity

The turbulence intensity for the 5 cases at the vertical distances $y=0.1\text{m}$, $y=0.2\text{m}$ and $y=0.3\text{m}$ are displayed in Figure 4.9. It can be drawn from Figure 4.9 (a) that the majority of the turbulence intensities are below 20% under various primary air pressures and supply air temperatures. In contrast with Figure 4.4, the lower velocity seems to have higher turbulence intensity, which supports the result of Melikov et al. [79]. Moreover, the area near the ceiling tends to have lower turbulence intensity, which is more clearly shown in higher primary air pressure (case 3). The reason could be that the lower air jet has stronger entrainment effects which results in higher turbulence intensity. Figure 4.9 (b) shows that the turbulence intensity distribution is different for non-isothermal cases. The turbulence intensity profiles are the reverse of the horizontal velocity profiles. Thus,

the temperature affects the turbulence intensity indirectly through the variation of velocity. This result aims for air flow in the ceiling region.

Figure 4.9 (a) Turbulence intensity for isothermal conditions (cases 1-3); (b) Turbulence intensity for non-isothermal conditions (cases 1, 4, 5); where y is the vertical distance from the ceiling

4.5 Summary

The air flow patterns of the air jet discharged by ACB system under isothermal and non-isothermal cases were fully investigated in this chapter. The velocity profiles along streamwise direction were crucial in analyzing the detachment of air jet from the ceiling. It was found out that the velocity would rise then decrease in streamwise direction. This streamwise peak velocity could be considered as a sign for the air jet to begin to detach from the ceiling. In addition, the higher primary air pressure contributed to a better Coandă effect, while the bigger differential temperature between the supply air and the room air hindered this effect. Therefore, in order to guarantee a good thermal comfort performance, the primary air pressure and the differential temperature should be carefully considered. The primary air pressure of 50Pa-150pa was suggested for ACB system and the differential temperature between the supply air and the room air above 4.7°C was not recommended. Although the air jet had self-similarity after discharged from the outlet, it was only feasible after that the air jet was developed and before vertical maximum air velocity went beyond 1.25 slot heights (0.05m). Furthermore, the self-similarity models were also unable to describe the air jet when the air jet was detached from the ceiling as the velocity profile was skewed. The mathematical model proposed by Verhoff and Schwarz & Cosart were compared with the experimental data. Overall, Verhoff's model was better than Schwarz and Cosart's as this model fit most of the experimental data. Moreover, the turbulence intensity at each testing point was also recorded and analyzed. The most of turbulent intensity of the air jet provided by the ACB system was within 20% under various primary air pressure s and supply air temperatures, which meant that ACB system could provide enough airflow rate and satisfied cooling performance with low turbulence intensity. The turbulence intensity tended to be larger for the airflow with lower velocity and stronger entrainment effect.

Chapter 5 Heat source effects on thermal comfort

5.1 Introduction

Thermal comfort is big challenge in modern society, where people spend most of their time in artificial buildings. Although some studies have characterized the effect of heat sources on the thermal comfort of chilled beam system, most of them are focused on passive chilled beam system [95, 96]. The research of heat sources' effects on ACB systems is not sufficient. As there is forced fresh air for ACB system, the airflow is not as vulnerable as passive chilled beam system. The effects of heat source may have different influences on ACB systems, which is well worth exploring. Besides, the current research of the heat sources' effects on thermal comfort pays more attention on the air distribution's effect such as draught problem [51, 53, 97]. The other characteristics, for example air temperature and humidity, are rarely taken into consideration. The humidity is of great importance in determining the sensitivity of occupants [98]. Jing et al. [99] proved through experiments that the high relative humidity in warm environment would cause uncomfortable sensation because the heat loss by evaporation was impeded. As thermal comfort is a comprehensive perception incorporating various factors, the overall full-scale evaluation of thermal comfort based on air temperature, speed and humidity are crucial. In addition, the thermal uniformity has significant effects on the modeling and evaluation of the occupied environment. In experimental research, the sensors shall be located according to the uniformity condition in order to describe the environment more accurately. In previous research, the thermal uniformity had been evaluated in terms of ADPI [66], which offered litter instructions on the sensor distribution.

In order to fill these gaps, the effects of the heat sources are experimentally investigated with ACB systems for summer seasons in this chapter. The full-scale distributions for air speed, temperature and humidity are mapped. The experimental setup is introduced in Section 5.2. The experiment results with respect to distribution of air speed, temperature

and humidity and evaluations of ADPI, PMV, DR, and VATD are presented in Section 5.3. The conclusions are summarized in Section 5.4

5.2 Experimental setup

5.2.1 The test room and measurement point distribution

The experiments are conducted in a well isolated mock-up room within an office building in Singapore. The dimension of this room is 7.3m*3.3m*2.5m (Length*Width*Height) (Figure 5.1). The outlet is grilled and located at the ceiling as indicated in Figure 5.2 with a dimension of 600mm*600mm. The occupied zone, which refers to the region normally occupied by people within a space, is generally considered to be zone between the floor and 1.8m above the floor and more than 1.0m from outside walls/windows and more than 0.3m from the internal walls [64]. Totally $9*5*4$ (x direction*y direction*z direction) testing points are evenly distributed in the occupied zone with the density of 0.5m (Figure 5.2). The heating panels (indicated in Figure 5.1) with various powers are adopted as the only heat sources for this room. The front side of the heating panels is made of carbon crystal, which will generate heat when energized. The back side of the heating panels is adiabatic to prevent heat conduction to the wall. A mobile robot equipped with the air speed, air temperature and relative humidity sensor collects all the data step by step. The movement of the mobile robot is based on the programmed instructions from the computer outside the mock-up room through wireless mode in order to keep good isolation of the room. The error for the position of the robot is $\pm 0.05\text{m}$. The sensors are set at the height of 0.1m, 0.6m, 1.1m and 1.7m based on the ASHRAE standard 113 [57].

Figure 5.1 The test room: 1- ACB; 2 – heating panels; 3- mobile robot

Figure 5.2 Testing point distribution

5.2.2 Measurement device

The air speed is tested by an omnidirectional air velocity transducer. The air temperature and the relative humidity are measured by the EE21 high-precision transmitter. The primary airflow rate is measured by a multifunction transmitter KIMO C310. The water temperature is measured by a high-precision thermistor. The water flow rate is measured by a turbine flowmeter. All the sensors are satisfied with the requirement of corresponding standards. The specifications for the air velocity transducer and multifunction transmitter are the same with those of sensors in Table 4.1. The information of the other sensors is shown in Table 5.1.

Table 5.1 Sensor specification

Sensor	Application	Type	Range	Accuracy	Standard
Humidity transmitter	Air relative humidity ratio	EE21	0~100%	$\pm 2\%$	Not specified in standards
Temperature transducer	Air temperature	EE21	-40~60°C	$\pm 0.2^\circ\text{C}$	EN 15116
Immersion temperature sensor	Water temperature	S1TH	-10~60°C	$\pm 0.1^\circ\text{C}$	ASHRAE Standard 200
Turbine flowmeter	Water flow rate	LWGY-A	0.1~0.6m ³ /h	$\pm 1\%$	ASHRAE Standard 200

5.2.3 Measurement procedures

In this chapter, two groups of experiments are investigated. Six cases are conducted as listed in Table 5.2. Case 1 is conducted under thermal absent condition with the same primary airflow rate with cases 2-4 as a base case for comparison. Cases 2-4 are to investigate the effects of the heating sources' configuration. They have the same primary air input but different heat sources' positions. Cases 4-6 are to evaluate the effects of the heat strength on the thermal comfort. The heat sources are located at the same positions under three different heat loads. Based on standard EN 15116 [34], the minimum cooling capacity is 15 W/m² floor area. The heat loads are suggested to be lower than 80 W/m² in [100] to ensure comfortable conditions in spaces. Based on these standards, three groups of heating panels with power of 400W, 800W and 1200 W, which correspond to the heat loads of 16.6 W/m², 33.2 W/m² and 49.8 W/m² respectively, are chosen in the experiment for comparison.

The parameters of the six cases are shown in Figure 5.3. In order to investigate the configurations' effect, three kinds of layouts for the heating panels are set up. For configuration I (Figure 5.3 (a)), the two heating panels are positioned non-symmetrically near the front and back walls. As regards to configuration II (Figure 5.3 (b)), the two

heating panels are set symmetrically near the middle of the front and back walls. As for configuration III (Figure 5.3 (c)), the two heating panels are set symmetrically near side walls. The heating panels are closed attached to the wall.

For cases 2-4, the effect of configurations is investigated under total thermal load of 800 W which is composed by two same heating panels. The same initial conditions (primary airflow rate and temperature, supply water flow rate and temperature) are imposed to these three cases. In order to make the ACB system the most energy efficient, the temperature of the chilled water should be as low as possible provided the condensation is avoided. Therefore, the supply water temperature is set at least 1°C higher than the dew point base on ASHRAE standards 55 [64]. The point at $x=3\text{m}$ and $y=0\text{m}$ is chosen as the reference point. The systems for cases 2-4 are considered to be stable when the reference point temperature for each case is varied within 0.2 °C for over one hour. Then the robot will be controlled to move to the test points. After stable, the robot will collect 20 groups of data of the air temperature, air speed and relative humidity ratio at a frequency of 1Hz. Ultimately the data will be transferred and stored to the computers outside the mock-up room.

The second test is to evaluate the effects of heat strength on the thermal comfort. Besides the 800 W heat load, another two heat loads 400 W and 1200W are also tested for configuration 3. In this test, the systems for cases 5 and 6 are considered stable when the reference point air temperature is kept at $\pm 0.3^\circ\text{C}$ with the temperature in case 4 for over one hour. Similarly, the air information will also be recorded. Experimental uncertainty analysis is shown in Table 5.3. As there are too many testing points for each case, it's not readable to show measuring uncertainty for each point. The maximum standard deviations of testing data are demonstrated.

Table 5.2 Experimental parameters

Cases	Heat load (W)	Configuration	Primary airflow rate (L/s)	Primary air temperature (°C)	Absolute humidity for primary air (g/m ³)	Supply water temperature(°C)	Water flow rate (L/h)
1	-	-	53.1	23.5	12.2	-	-
2	800	1	53.1	19.2	12.2	14.6	118
3	800	2	53.1	19.2	12.2	14.6	118
4	800	3	53.1	19.2	12.2	14.6	118
5	1200	3	71.1	17.1	12.2	14.0	164
6	400	3	32.2	21.5	12.2	14.5	83

Table 5.3 Experimental uncertainty analysis

cases	Maximum velocity standard deviation	Maximum temperature standard deviation	Maximum humidity standard deviation
1	0.087	0.083	0.041
2	0.112	0.071	0.066
3	0.097	0.076	0.045
4	0.131	0.096	0.048
5	0.067	0.087	0.04

Figure 5.3 Configurations of heating panels: (a) configuration I; (b) configuration II; (c) configuration III

5.3 Results

The air speed distribution is one the most important factors for air diffusion condition and thermal comfort performance. The accurate understanding of the air speed distribution can be referred to offer good installation and ventilation schemes. The temperature and humidity for the occupied zone are normally represented by the average value for several positions or just a particular position. The data are significantly affected by the location of the sensors. Through the analysis of the air temperature and humidity distribution and uniformity, the most effective and economical sensor distribution scheme can be confirmed. Therefore, the air temperature, air speed and relative humidity for 160 points are measured in the occupied zone and 20 groups of data are collected for each point over

a period of 20s. The average values are calculated and adopted to represent the value at the particular point. The full-scale thermal distributions are profiled and the uniformities are evaluated. Thermal comfort is evaluated by indices of ADPI, PMV, DR and VATD for cases 2-6. Case 1 is eliminated as no heat source is applied.

5.3.1 Air speed distribution

The average air speeds of all test points for the cases 1-6 are 0.19m/s, 0.25m/s, 0.23m/s, 0.24m/s, 0.3m/s and 0.2m/s, respectively. Although cases 1-4 have the same primary airflow rate, the average air speed for thermal absent case 1 is much smaller than the other three cases. The average air speeds for cases 2-4 are quite close. For cases 4-6, the average speeds are positive correlated with the supply airflow rates.

The air speed distribution for all cases in the occupied places is depicted in Figure 5.4. ACB terminal is located in the middle of the measured zone, which is between 1.7m-2.3m (x direction) and 0.4m-1.6m (y direction). It can be observed that the air speeds at 1.7m, especially on the two sides of the ACB terminal unit, are distinctly higher than other positions for all cases. While, most of the air speeds for 1.1m (sedentary neck level) are within 0.35m/s, which is acceptable in terms of thermal performance. It reveals that the ACB terminals can make a good advantage of Coandă effect that the supply air does not blow to the sedentary occupants directly. The air speed is much smaller for thermal absent case compared with cases 2-4 under the same primary airflow rate. It can be drawn that the heat sources can enlarge the air speed in the occupied zone, which is in accordance with the result of Kosonen et al. [53]. For cases 2-4, the speed distribution has no obvious difference with each other for higher height above 0.6m. The air speed at 0.1m for case 4 is a little bit larger than case 2 and case 3, which may due to the thermal plume caused by the difference of heat source configuration. It is concluded that the heat sources can enhance the average air speed, nevertheless, the configuration of the heat sources has little effect to the air speed distribution for primary airflow rate 32.2 L/s. As for the effect of the heat strength (cases 4-6), the average air speed and maximum speed are larger when the supply airflow rates are stronger to balance the extra heat loads.

Figure 5.4 (a)-(f): Air speed distribution for cases 1-6, respectively

5.3.2 Air temperature distribution

The average air temperatures of all test points for the cases 1-6 are 23.5°C, 24.3°C, 24.6°C, 23.9°C, 23.7°C and 23.7°C, respectively. Although the initial input is same for cases 2-4, the different configuration results in different average temperatures. For cases 4-6, as the heat load is different, the zone temperature is kept at the same.

The air temperature distribution is shown in Figure 5.5. The air temperature distribution is quite uniform for all heights for case 1, which demonstrates the air temperature distribution when there are no heat sources. The air temperatures at 1.7m are slightly lower than the temperature at other heights for cases 2-6, which is due to that the height 1.7m is too close to cooling supply air from the ACB unit. However, the temperatures at height 1.1m are slightly higher than the other heights based on the observations of cases 4-6. As the ACB unit has good Coandă effect, the supply airflow is attached to the ceiling after being discharged from the terminal unit. Then the cooling air will descend after hitting the wall. Therefore, the temperatures in the middle height (1.1m and 0.6m) are not directly affected by the supply air. For cases 2-4, as the different configuration of the heat sources, the temperature distribution is totally different. For case 2, the air temperature distribution is extremely non-uniform as the air temperature is higher when it is closer to the heat sources. However, the temperatures are more uniform for case 3 and case 4 compared with case 2, when the heating panels are placed symmetrically. The average temperature for case 3 is higher than that for case 4 as the heating panels are closer to the occupied zone. For cases 3, 4 and 5, as the configurations of the heating panels are the same, the temperature distributions are all relatively the same.

Figure 5.5 (a)-(f): Air temperature distribution for cases 1-6

5.3.3 Air humidity distribution

In this study, the relative humidity distribution is also tested for all cases. The result turns out that the distribution of the relative humidity is exactly contrary to the distribution of the temperature. As the relative humidity is inversely related to the temperature, it fails to describe the authentic humidity condition when the air temperature is different. Based on the results from Figure 5.5, the temperature varies different at different positions. Thus, the relative humidity distribution is not reliable. Thus, the absolute humidity for each point is calculated based on the following equation [101]:

$$P_{ws} = A \cdot 10^{\left(\frac{m \cdot T}{T + T_n}\right)} \quad (5.1)$$

where P_{ws} is the saturated water vapor pressure (hPa); T is the temperature (°C); A , m and T_n are constants, which are 6.116, 7.59 and 240.7 in this experiment, respectively.

$$RH = P_w / P_{ws} \cdot 100\% \quad (5.2)$$

where RH is relative humidity (%); P_w is the water vapor pressure.

$$AH = P_w / (R_w \cdot T) \quad (5.3)$$

where AH is the absolute humidity(kg/m³); R_w is constant number, which is 461.5 J/(kg·K); T is the air temperature (K).

The absolute humidity distribution is indicated in Figure 5.6. The average absolute humidity for cases 1-6 is 14.17 g/m³, 12.53 g/m³, 11.17 g/m³, 11.24 g/m³, 11.12 g/m³ and 12.15 g/m³. The overall absolute humidity distributions for all cases are quite uniform. The absolute humidity is significantly higher than the other cases when there is no heat source (as shown in Figure 5.6(a)). For cases 2-4, no obvious relationships can be found for the absolute humidity and heat sources' location. For cases 4-6, the higher the heat load, the smaller the average absolute humidity. It can be inferred that heat sources can notably affect the humidity condition in the occupied zone. Other factors such as the

location of the humidity source and initial humidity condition may also have effect on the humidity distribution, which are not studied in this chapter.

Figure 5.6 (a)-(f): Absolute humidity distribution for cases 1-6

5.3.4 Uniformity

The smaller the standard deviation means the better uniformity for the data. The results are shown in Figure 5.7. It can be seen from Figure 5.7(a) that all cases have good humidity performance regardless the heat sources' location and heat strength. The temperature distribution is quite uniform when there is no heat source inside the mock-up room (case1). Cases 4-6 all have a relatively better temperature uniformity compared with case 2 and case 3, which means configuration III has best performance on the temperature uniformity. For absolute humidity, the configuration has little effect on the humidity distribution. Although the strong heat strength could reduce the absolute humidity value, the uniformity is not affected. When it comes to the air speed, it is observed that the uniformity is related with the supply airflow rate. For cases 2-4, which have the same primary air pressure, the speed uniformities are close to each other. Through the comparison for cases 4-6, it can be seen that the high supply airflow rate will deteriorate the uniformity of speed.

In association with the distribution (Figure 5.5 and Figure 5.6) and uniformity of air temperature and humidity (Figure 5.7), the effective and economical sensor scenarios can be determined. It is shown that the standard deviations of absolute humidity for all cases (Figure 5.7 (a)) and all heights (Figure 5.7 (b)) are within 0.1, which means the humidity are quite uniform for all the positions under various cases. It can also be observed clearly from (Figure 5.6). Based on the analyses above, one humidity sensor is suitable to measure the zone humidity for ACB system under various heat source conditions. For symmetric heat sources (cases 3-6), the temperatures at height 1.7m are slightly lower than the other heights (Figure 5.5) and the standard deviations at 1.7m are a little bit higher than that of other positions (Figure 5.7 (c)). However, the temperatures are quite uniform for heights 0.1m-1.1m. Therefore, two temperature sensors can be set up for heights above 1.7m and under 1.1m. The average temperature can be used as the zone temperature. The temperature distribution is quite non-uniform when the heat sources are non-symmetrically located (case 2). In this case, more temperature sensors shall be set up closely and away from the heat sources.

Figure 5.7 (a): Uniformities of air temperature, speed and absolute humidity; (b):Uniformities of air absolute humidity for each height; (c):Uniformities of air temperatures for each height;

5.3.5 ADPI

As this index applies to the sedentary behaviors, which refer to the activity in a sitting reclining or lying posture with no more than 1.5 metabolic equivalents [102]. Therefore, the ADPI are characterized for height 0.1m, 0.6m and 1.1m. The results are depicted in Figure 5.8. The ADPI indices for case 2-6 are 93.3%, 97.0%, 89.6%, 74.1% and 99.3%, respectively. It indicates that configuration II (Figure 5.8 (b)) obtains the best performance among cases 2-4 for room average temperature around 24 °C. However, when the room temperature is higher, the air speed can be elevated to resist the warm environment. The best configuration for ADPI performance may be different. For cases 4-6, case 5 has the worst performance and case 6 has the best ADPI value. The points are beyond the criteria ($-1.7^{\circ}\text{C} < T_{ed} < +1.1^{\circ}\text{C}$ and $v < 0.35\text{m/s}$) mostly because air speed is larger than 0.35m/s and the majority of them are distributed at $h=0.1\text{m}$. For higher heat strength, the supply airflow rate is elevated to balance the heat load, which will lead to the excessive air speeds in the occupied zone. Although one ACB terminal can achieve 1200W heat load, however, the thermal comfort is deteriorated because of the large air speed. As the cooling capacity of the induced air side has reached maximum, more ACB terminals can be applied when the heat load is larger than 1200W to maintain the thermal comfort.

Figure 5.8 (a)-(e): ADPI results for cases 2-6

5.3.6 Predicted mean vote (PMV)

In this experiment, the PMV is evaluated for sedentary condition whose metabolic rate is set as 1.2 met. The effective mechanical power is set as 0. The mean radiant temperature is assumed as same as the mean air temperature. As the general office room temperature is around 22.5-25.5 °C in Singapore [103], the typical garment condition is panties, short sleeves, light trousers, socks, shoes (thin soled) and a light jacket. The clothing insulation is totally 0.67 clo based on the ISO 7730.

The results are sketched in Figure 5.9 for height 0.1m, 0.6m and 1.1m, which represent the heights of ankle, thigh and neck for occupants in sedentary condition. The ranges of PMV for cases 2-6 are -0.29~0.26, -0.14~0.27, -0.42~0.09, -0.57~-0.02, and -0.34~0.19, respectively. In general, ACB terminal can provide a good thermal comfort as the majority of the PMV values are within ± 0.5 , which is credited as category B in in ISO 7730. It can be seen that the air speed has an overwhelming negative effect to the PMV. The large air speed appears mostly at the ankle level (height 0.1m). Thus, the ankle level has a higher risk to feel cold. The thigh and neck levels appear to have better thermal comfort as most of the PMV are between ± 0.2 (category A in ISO 7730). For cases 2-4, the PMV values are not significantly affected by heat sources' configuration. The higher heat strength will lower the PMV due to the excessive air speed (Figure 5.9 (d)).

Figure 5.9 (a)-(e): PMV values for cases 2-6

5.3.7 Draught rate & Vertical air temperature difference

The maximum vertical air temperatures differences (VATD) for the testing points at 0.1m and 1.1m are calculated and depicted in Table 5.4. It can be observed that maximum VATD is no more than 1°C, which highly meets the requirement of 2.7°C in the standards 7730 for category A.

The overall thermal comfort performance has been summarized in Table 5.4. As cases 3 and 4 have the better thermal comfort performances (category B) compared with case 2 (category C), it can be concluded that the symmetrical heat source's configuration can provide more satisfied thermal condition. As for the strength of heat sources, case 5 behaves the worst performance (category C) among cases 4-6. When the heat strength is too high, the thermal comfort is difficult to be maintained.

Table 5.4 Thermal comfort performance

Cases	ADPI (%)	PMV	DR (%)	Maximum VATD between h=0.1m and h=1.1m (°C)	Category
2	93.3	-0.26~0.26	21.5	0.30	C
3	97.0	-0.14~0.27	17.8	0.12	B
4	89.6	-0.44~0.09	18.1	0.23	B
5	74.1	-0.57~-0.02	22.1	0.24	C
6	99.3	-0.34~0.19	19.2	0.35	B

5.4 Summary

The effects of heat sources' configuration and heat strength on thermal comfort have been evaluated through experiments in a mock-up room in Singapore. The air temperature, air speed and absolute humidity distribution have been mapped systematically. The corresponding uniformities have been characterized. The results show that ACB system can provide good Coandă effects under various conditions so that the sedentary zone (0.1m-1.1m) are not directly influenced by the supply airflow. Hence,

the uncomfortable draught is reduced and a good thermal comfort environment is ensured. The temperature distribution is relatively uniform when the heat sources are symmetrical. The average value of two temperatures sensors (one is above 1.7m and the other one is below 1.1m) can be utilized to represent the zone temperature. As the absolute humidity distribution is extremely uniform for all cases, one humidity sensor can be applied to measure the zone humidity for ACB systems when there is no significant humidity source. It should be noted that the relative humidity distribution cannot be applied when the temperature distribution is not uniform. This is due to that the relative humidity varies under different temperatures even though the absolute humidity is same. The thermal comfort performances are evaluated by the indices of ADPI, PMV, DR and VATD. The results turn out that the symmetrical heat sources have better thermal comfort performance than non-symmetrical heat sources. The thermal comfort performance will be deteriorated by the excessive air speed to reject the extra heat load. The heat loads are suggested to be no more than 1200 W for one ACB terminal unit applied in order to maintain a good thermal comfort environment.

Chapter 6 Optimization for the ACB systems

6.1 Introduction

After thoroughly investigation on thermal comfort, practical guidelines are achieved on operating ACB systems. It should be noted that those operations shall be at reasonable energy cost. Thus, accurate models and appropriate optimization strategies for ACB systems are indispensable. However, little of previous research addresses the optimization strategies for ACB systems. This chapter makes some progress by proposing an optimization strategy with the genetic algorithm (GA) to improve the performance of ACB systems with respect to energy consumption and thermal comfort.

Section 6.2 proposed the models encapsulating the energy consumption and heat transfer rate. The optimization strategy is elaborated in Section 6.3. The experimental setup and model validation are illustrated in Section 6.4. The optimization results and the original experiment results are compared in Section 6.5. Eventually, the conclusions are drawn in Section 6.6.

6.2 Model development

In order to well describe the performance of ACB systems, models for each parts of the system have to be built accurately. The energy consumption models can be divided into chiller model, pump model and fan model. As there is a coil inside ACB terminal to cool the induced air, a heat transfer model for coil is also necessary. The widely acknowledged thermal comfort model of PPD is adopted to evaluate the thermal comfort performance of ACB systems.

6.2.1 Chiller model

There are two chillers in ACB systems. One chiller provides cooling for primary air while the other offers cooling for the chilled water in the cooling coil. The power

consumption of a chiller system can be estimated in terms of the part load ratio which refers to the ratio of the current load and the designed capacity of the chiller [104].

The current load of chilled water is determined by the cooling requirement which can be expressed as follows:

$$Q_{w,cur} = c_w \dot{m}_w (T_{w,in} - T_{w,out}) \quad (6.1)$$

where $Q_{w,cur}$ is the current load of chiller for chilled water; c_w is the specific heat of chilled water; $\dot{m}_w$ is the mass flow rate of chilled water; $T_{w,in}$ and $T_{w,out}$ are the temperatures of inlet and outlet of chilled water.

The current load of the primary air is expressed of its enthalpy as:

$$Q_{a,cur} = \dot{m}_a (h_{a,in} - h_{a,out}) \quad (6.2)$$

where $Q_{a,cur}$ is the current load of chiller for primary air; $\dot{m}_a$ is the mass flow rate of primary air; $h_{a,out}$ is the enthalpy of outdoor air; $h_{a,in}$ is the enthalpy of primary air.

The part load ratio of chilled water can be represented as:

$$PLR_w = \frac{Q_{w,cur}}{Q_{c,nom}} \quad (6.3)$$

where PLR_w is the part load ratio of chilled water; $Q_{c,nom}$ is the nominal cooling capacity for chiller.

Similarly, the part load ratio of primary air is:

$$PLR_a = \frac{Q_{a,cur}}{Q_{c,nom}} \quad (6.4)$$

where PLR_a is the part load ratio of primary air.

The chiller power consumption for the chilled water can be calculated in terms of part load ratio as:

$$E_{w,c} = a_{w,3} PLR_w^3 + a_{w,2} PLR_w^2 + a_{w,1} PLR_w + a_{w,0} \quad (6.5)$$

where $a_{w,0} \sim a_{w,3}$ are the parameters that fitted by the experiments.

The chiller power consumption for the primary air is calculated as:

$$E_{a,c} = a_{a,3} PLR_a^3 + a_{a,2} PLR_a^2 + a_{a,1} PLR_a + a_{a,0} \quad (6.6)$$

where $a_{a,0} \sim a_{a,3}$ are the parameters that fitted by the measurement data from experiments.

6.2.2 Model of fan and pump

The power consumption of fan and pump can be estimated as a cubic function of the ratio of the solution flow rate to the nominal flow rate and the nominal power consumption as follow [105]:

$$E_p = E_{p,nom} (a_{p,3} (\dot{m}_w / \dot{m}_{w,nom})^3 + a_{p,2} (\dot{m}_w / \dot{m}_{w,nom})^2 + a_{p,1} (\dot{m}_w / \dot{m}_{w,nom}) + a_{p,0}) \quad (6.7)$$

$$E_f = E_{f,nom} (a_{f,3} (\dot{m}_a / \dot{m}_{a,nom})^3 + a_{f,2} (\dot{m}_a / \dot{m}_{a,nom})^2 + a_{f,1} (\dot{m}_a / \dot{m}_{a,nom}) + a_{f,0}) \quad (6.8)$$

where $E_{p,nom}$ and $E_{f,nom}$ are the nominal power consumption for pump and fan, respectively; $\dot{m}_{w,nom}$ and $\dot{m}_{a,nom}$ are the nominal mass flow rate for chilled water and primary air; $a_{f,0} \sim a_{f,3}$ and $a_{p,0} \sim a_{p,3}$ are the fitted parameters by the operating data.

6.2.3 Heat transfer model

According to ASHRAE standard 200, the coil heat transfer coefficient (K) can be described by the following empirical equation:

$$K = a(\nu\rho_i)^{n1} \omega^{n2} \quad (6.9)$$

where $\nu\rho_i$ is the mass velocity of induced air, ω is the velocity of water in the coil; $n1$ and $n2$ are empirical parameters.

Assume the induced air density and IR are constant numbers. The cross Section of the duct and coil keep same. This heat transfer rate of the coil can be changed to the following equation:

$$Q_h = a(IR \cdot \dot{V}_p)^{n1} \dot{V}_w^{n2} (T_{zone} - T_w) \quad (6.10)$$

where Q_h is the heat transfer rate of the coil; $\dot{V}_p$ is primary airflow rate; $\dot{V}_w$ is chilled water flow rate; T_{zone} is the temperature of the occupied zone; T_w is the temperature of the chilled water.

6.2.4 Thermal comfort model

The thermal comfort is represented by predicted percentage dissatisfied (PPD), which is an expression of predicted mean vote (PMV). The smaller PPD value means better performance of thermal comfort. The calculation of PMV and PPD can be referred in Chapter 2. In this model, the PMV is calculated for sedentary condition with metabolic rate of 1.2 met. The effective mechanical power is set as 0. The mean radiant temperature is assumed as same as the mean air temperature. The air temperature and humidity are based on the requirement in the occupied zone. The maximum air velocity in the zone is adopted as the relative air velocity in this model. The maximum air velocity is found to have a nonlinear relationship with the primary airflow rate (as shown in Eq. (6.11)).

$$V_{\max,a} = c1 \cdot \dot{V}_p^{c2} \quad (6.11)$$

where $V_{\max,a}$ is the maximum air velocity in the occupied zone; $c1$ and $c2$ are empirical parameters.

6.3 Global optimization strategy

6.3.1 Objective function

The total energy consumption for this ACB system is the sum power of the fan, the pump, and chillers.

$$E_{total} = E_f + E_{w,c} + E_{a,c} + E_p \quad (6.12)$$

where E_{total} is the total power consumption; E_f is the power consumption by fan; $E_{w,c}$ is the power consumption by chiller of chilled water; $E_{a,c}$ is the power consumption by chiller of primary air; E_p is the power consumption by pump.

The primary object of the optimization strategy is to provide the conditioned space with required cooling capacity while minimizing the total energy consumption and thermal discomfort. Therefore, the objective function is established as follows:

$$\min J = \lambda E_{total} + (1 - \lambda)PPD \quad (6.13)$$

where λ is the weight factor.

As the magnitude of energy consumption is much larger than PPD, the energy consumption is normalized to 0-1. PPD is not normalized as it is naturally from 0.05 to 1. The objective function is updated as:

$$\min J = \lambda E_{nor} + (1 - \lambda)PPD \quad (6.14)$$

6.3.2 Constrains

Due to the limitations of the equipment and the practical operating conditions, some constrains are imposed.

The chiller load is restricted by its capacity as follows:

$$Q_{chiller,a,min} < Q_{chiller,a} < Q_{chiller,a,max} \quad (6.15)$$

$$Q_{chiller,w,min} < Q_{chiller,w} < Q_{chiller,w,max} \quad (6.16)$$

The minimal primary airflow rate is restricted by the ventilation rate based on ASHARE standard 62.1 and maximum primary airflow rate is restricted by the maximum fan power.

$$\dot{V}_{p,min} < \dot{V}_p < \dot{V}_{p,max} \quad (6.17)$$

The water flow rate is limited by the water pump power:

$$\dot{V}_{w,min} < \dot{V}_w < \dot{V}_{w,max} \quad (6.18)$$

In steady condition, the total cooling capacity provided by the primary air and cooling coil in ACB terminal shall be equal to the cooling load in the occupied zone.

$$Q_p + Q_h = Q_{zone} \quad (6.19)$$

$$Q_p = \dot{m}_a (h_{a,in} - h_{zone}) \quad (6.20)$$

where Q_{zone} is the cooling load of the occupied zone; Q_p is the cooling capacity of primary air; h_{zone} is the enthalpy of the occupied zone.

When the system is stable, the heat transfer rate of the coil is considered be equal to the cooling capacity of chilled water:

$$Q_h = Q_{w,cur} \quad (6.21)$$

The zone temperature, zone cooling load, enthalpy of the zone, chilled water temperature, and enthalpy of primary air depend on the requirements. The enthalpy of outdoor air is measured by the sensors.

$$T_{zone} = T_{req,zone} \quad (6.22)$$

$$h_{zone} = h_{req,zone} \quad (6.23)$$

$$Q_{zone} = Q_{req,zone} \quad (6.24)$$

$$h_{a,in} = h_{req,a,in} \quad (6.25)$$

$$h_{a,out} = h_{mea,a,out} \quad (6.26)$$

$$T_w = T_{req,w} \quad (6.27)$$

The controllable parameters are $\dot{V}_p$ and $\dot{V}_w$, which will be optimized to achieve the goal of minimizing the energy consumption and thermal discomfort.

The optimization model can be summarized as follows:

$$\begin{aligned} \min \quad & J = \lambda E_{nor} + (1 - \lambda) PPD \\ \text{subject to: } & Q_{chiller,a,min} < Q_{chiller,a} < Q_{chiller,a,max} \\ & Q_{chiller,w,min} < Q_{chiller,w} < Q_{chiller,w,max} \\ & \dot{V}_{p,min} < \dot{V}_p < \dot{V}_{p,max} \\ & \dot{V}_{w,min} < \dot{V}_w < \dot{V}_{w,max} \\ & Q_{zone} = Q_{req,zone} \\ & T_{zone} = T_{req,zone} \\ & h_{zone} = h_{req,zone} \\ & h_{a,in} = h_{req,a,in} \\ & h_{a,out} = h_{rea,a,out} \\ & T_w = T_{req,w} \end{aligned} \quad (6.28)$$

6.3.3 Genetic algorithm optimization strategy

The genetic algorithm (GA) is adopted to solve this optimization problem. GA is an uncertain searching method based on natural selection in evolution process. The logic is to eliminate the less adaptive population and keep the population with high adaptability to the environment. The left population will generate more offspring through selection, crossover, and mutation. A GA optimization strategy for energy consumption and thermal comfort can be divided into the following steps:

Step 1: Determine the parameters T_{zone} , h_{zone} , Q_{zone} , $h_{a,in}$, $h_{a,out}$ and T_w based on the space condition and comfort requirement.

Step 2: Get the limitations of the chiller load, primary airflow rate and chilled water flow rate through practical operating condition. Calculate constrains for GA.

Step 3: Initialize the population randomly for primary airflow rate ($\dot{V}_p$) and chilled water flow rate ($\dot{V}_w$). The initial population will be coded into binary strings, which are also called chromosomes in biology.

Step 4: Calculate the fitness value for each chromosome in initial population based on the fitness function, which refers to the objective function in this chapter.

Step 5: Generate the offspring by carrying out selection, crossover and mutation for the chromosome. Selection is to generate a new population made of chromosomes with high fitness value based on the method of roulette wheel. The chromosomes will be ranked according to their fitness value. The larger fitness value has higher probability to be selected. Crossover is to exchange parts of chromosomes under a pre-determined crossover rate. Mutation is to reverse a random digit for certain chromosomes based on pre-determined mutation probability.

Step 6: Repeat steps 2 and 3 until iteration steps are satisfied or the fitness value does not change any more. The maximum fitness value is recorded. Then corresponding chromosomes are decoded and $\dot{V}_p$ and $\dot{V}_w$ are identified.

The flow chart of the optimization strategy is depicted in Figure 6.1. The settings of GA in this chapter are illustrated in Table 6.1.

Table 6.1 GA settings

Parameters	Value
Population size	50
Maximum generation	50
Probability of mutation	0.01
Probability of crossover	0.8
Selection	Roulette wheel selection

Figure 6.1 Flow chart of the optimization strategy

6.4 Experimental setup and model validation

6.4.1 Experimental setup

In order to demonstrate the proposed GA optimization method, a typical ACB system including two chillers, a fan, an air handling unit (AHU), heat exchangers, pumps, and a mock-up room with one ACB terminal are set up. A chiller mainly consists of a compressor, a condenser, an evaporator, a pump, and an electric valve. The characteristics for chillers 1 and 2 are the same. The schematic diagram is illustrated in Figure 6.2. The experimental facilities are shown in Figure 6.3. The data of temperature, humidity, flow rate, and air velocity are integrated and registered by a LabVIEW program depicted in Figure 6.4. The components nominal capacities are shown in Table 6.2.

Figure 6.2 Schematic diagram of the ACB system

Figure 6.3 Experimental facilities for the ACB system: 1: AHU; 2: Fan; 3: Water tank; 4: Water pump for chiller 1; 5: Water pump for chiller 2; 6: Water pump for coil in ACB terminal; 7: Compressor for chiller 1; 8: Compressor for chiller 2; 9: Evaporator for chiller 1; 10: Evaporator for chiller 2; 11: Condenser for chiller 1; 12: Condenser for chiller 2; 13: Control cabinet

Figure 6.4 LabView program

Table 6.2 Components nominal capacities

Component	Nominal capacity (W)	Nominal flow rate (m3/h)
Chiller	4000 W	-
Pump	370 W	2.0795
Fan	300 W	243

6.4.2 Model validation

To evaluate those models, room mean square of relative error (RMS_RE) and standard deviation of relative error are (STD_RE) are adopted.

$$RE_i = \frac{|\tilde{y}_i - y_i|}{y_i} \times 100\% \quad (6.29)$$

$$RMS_RE = \sqrt{\frac{\sum_{i=1}^M (RE_i)^2}{M}} \quad (6.30)$$

$$MRE = \frac{\sum_{i=1}^M RE_i}{M} \quad (6.31)$$

$$STD_RE = \sqrt{\frac{\sum_{i=1}^M (RE_i - MRE)^2}{M - 1}} \quad (6.32)$$

The proposed models for the fan, the pump, chillers, heat transfer rate, and maximum velocity are validated through experiments. The predicted and measured values are compared from Figure 6.5 to Figure 6.10, which shows a good agreement with relative errors within 10% for heat transfer model and 5% for other models. The accuracy of each model is evaluated by RMS_RE and STD_RE. The results are demonstrated in Table 6.3.

Figure 6.5 Model validation for the fan energy model

Figure 6.6 Model validation for pump energy model

Figure 6.7 Model validation for chiller 1 energy model

Figure 6.8 Model validation for chiller 2 energy model

Figure 6.9 Model validation for heat transfer model of coil in ACB terminal

Figure 6.10 Model validation for maximum air velocity in the occupied zone

Table 6.3 Accuracy of proposed models

Models	RMS_RE	STD_RE
Chiller 1 model	0.0114	0.0072
Chiller 2 model	0.0143	0.0109
Pump model	0.0134	0.0092
Fan model	0.0138	0.0101
Heat transfer model	0.0377	0.0224
Maximum velocity	0.0332	0.0199

6.5 Optimization results

As this optimization strategy is based on stable condition, three stable cases are tested according to empirical operations. The parameters for the three cases are illustrated in Table 6.4. The energy consumption and PPD are compared for the optimization results and original results in Figure 6.11 and Figure 6.12, respectively. With the optimization method, the energy is saved 2240.9 W, 915 W, and 57.1 W for case 1, case 2 and case 3 respectively. The PPD is reduced by 4.14 %, 0.83 %, and 0 % for case 1, case 2 and case 3 respectively.

Table 6.4 Three original cases

Cases	Primary air enthalpy (kJ/kg)	Chilled water temperature (°C)	Stable zone temperature (°C)
1	37.62	14	23.5
2	41.52	14.6	23.9
3	47.57	14.6	23.9

Figure 6.11 The original and optimization results for energy consumption

Figure 6.12 The original and optimization results for PPD

The weight factor λ in the cost function is evaluated in terms of case 1. It can be seen from Figure 6.13 and Table 6.5 that when λ rises from 0 to 0.5, energy consumption is decreased by 349.8 W and PPD increased by 0.05%. When λ continues rising, both energy consumption and PPD have little changes. One reason is that although energy consumption is normalized from 0 to 1, it is still around 10 times of PPD. The other reason is that the minimal PPD is 5% and the optimized PPD is already small enough. Therefore it cannot be further reduced.

Figure 6.13 Effects of weight factor λ

Table 6.5 Effects of weight factor λ

λ	Energy consumption (KW)	PPD (%)
0	6.7223	5.23%
0.2	6.3734	5.28%
0.5	6.3725	5.29%
0.8	6.3724	5.29%
1	6.3724	5.29%

6.6 Summary

In this chapter, an optimization strategy is proposed for ACB systems based on genetic algorithm. The energy consumption and thermal comfort is optimized by adjusting the primary airflow rate and water flow rate. Energy models for the fan, pump, chillers, heat transfer model for the coil in ACB terminal, and velocity model are developed and validated by experiments. The weight factor λ is evaluated. The value of 0.5 is suggested for λ to achieve the best performance for both energy consumption and thermal comfort. Three experimental cases based on the empirical operation are compared with the optimized strategy. The results demonstrate that the proposed optimization method offers a better performance for energy consumption and thermal comfort.

Chapter 7 Conclusions and future work

7.1 Conclusions

This thesis mainly studies the performances of ACB systems with respect to air distribution, thermal comfort, and energy consumption numerically and experimentally. It can be concluded that ACB systems overall can provide a good thermal comfort performance under various operating conditions. However, there is still huge potential for ACB systems to be improved.

A un-uniformity of air distribution is found near ACB nozzles and the effect is proved to be significant when the nozzle diameter is large. Moreover, the rise of the primary air pressure will aggravate this un-uniformity. This problem can be solved by adding a board in the middle of primary air plenum of ACB terminal. With this board, the air will be discharged evenly from the nozzles.

The characteristics of air jet are closely related to the supply air temperature and primary air pressure. The high primary air pressure will delay the detachment of air jet from the ceiling, while the low supply air temperature will have the opposite effects. This streamwise peak velocity can be utilized to predict the detachment from the ceiling.

The self-similarity of air jet is found to be feasible after that the air jet is developed and before vertical maximum air velocity goes beyond 1.25 slot heights (0.05m) for the ACB systems. The self-similarity models fail to describe the air jet when the air jet is detached from the ceiling. Besides, most of turbulent intensities for the air jet provided by the ACB systems are within 20%. The turbulence intensity tends to be larger for the airflow with lower velocity and stronger entrainment effect.

The heat sources have significant effects on thermal comfort. It turns out that the symmetrical heat sources have better thermal comfort performance than non-symmetrical heat sources. The thermal comfort performance will be deteriorated by the excessive air

speed to reject the extra heat loads. Therefore, the heat loads are suggested to be no more than 1200 W for one ACB terminal in order to maintain a good thermal comfort environment. The absolute humidity distribution is actually quite uniform for all cases. One humidity sensor can be applied to measure the zone humidity for ACB systems when there is no significant humidity source. The temperature distribution depends on the heat sources and supply air. Normally, the temperature at height 1.7m is lower than that of other heights. The air speeds at 1.7m especially on the two sides of the ACB terminal unit, are distinctly higher than other positions for all cases. While, most of the air speeds for 1.1m (sedentary neck level) are within 0.35m/s, which is acceptable in terms of thermal performance. Air speed has an overwhelming negative effect to the PMV. The large air speed appears mostly at the ankle level (height 0.1m) for sedentary activity. Thus, the ankle level has a higher risk to feel cold. The maximum VATD is no more than 1°C, which highly meets the requirement of 2.7°C in the standards 7730 for category A.

The performance of energy consumption and thermal comfort can be improved simultaneously by employing optimization methods. The weight factor λ has some effects on the optimization results. However, the effect is not significant as the PPD is still very small compared with the normalized energy consumption.

7.2 Future work

Although many efforts have been devoted to study the ACB systems, further investigation is still essential to improve the performance of ACB systems.

In this thesis, the airflow model only addresses in ACB terminal and near the ceilings. As discussed in conclusion, the air speed has an overwhelming effect on thermal comfort compared with temperature and humidity. The airflow pattern in the occupied zone can be further studied in the future. Accurate models can be established to predict the magnitude of air speed. The directions of the airflow may also have effects on thermal comfort. Other measurement methods, for example particle image velocimetry (PIV), can be utilized to evaluate the effects of direction of airflow.

In addition to the air distribution, the effects of heat sources are mainly evaluated for thermal comfort of ACB systems in this thesis. Other factors, such as the space of the occupied zone, arrangements of ACB, outdoor conditions, interactions of adjacent rooms, human activities can also be taken into considerations.

Despite GA method can provide satisfactory results for the whole system optimization, it needs many generations to locate the optimal set point. Moreover, Eq.(2.7) needs iterations to solve the solution in the model of PPD. It is very time-consuming to use GA method for optimizing the PPD. In the future, more effective and efficient algorithm will be explored to optimize the ACB systems.

As the condensation is strictly prohibited for the coil inside ACB terminal, a dehumidification system is necessary for areas with high humidity ratio. More energy will be consumed on the process of removing water vapor from the outdoor air. The optimization strategy for energy consumption will be different from what is proposed in this thesis, which is well worth studying in the future.

Author's Publications

Wu B, Cai W, and Ji K. "Heat source effects on thermal comfort for active chilled beam systems," Building and Environment. vol .141, pp.91-102, 2018.

Wu B, Cai W, Chen H, and Ji K. "Experimental investigation on airflow pattern for active chilled beam system," Energy and Buildings. vol.166, pp.438-49, 2018.

Wu, B., Cai, W., Wang, Q., Chen, C., Lin, C. and Chen, H.. "The air distribution around nozzles based on active chilled beam". International High Performance Buildings Conference, 2016.

Wu B, Cai W, and Chen H. "Effects of airflow rate and supply air temperature on the airflow pattern under active chilled beam system," 13th International Conference on Industrial Electronics and Applications (ICIEA), 2018.

References

- [1] "<https://en.wikipedia.org/wiki/HVAC>," *Wikipedia*.
- [2] 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, 2014.
- [3] U. Berardi, "Building energy consumption in US, EU, and BRIC countries," *Procedia engineering*, vol. 118, pp. 128-136, 2015.
- [4] IPCC, "Intergovernmental Panel on Climate Change, Climate Change 2014: Mitigation of Climate Change, Chapter 9: Buildings," 2014.
- [5] D. G. Fridley, N. Zheng, and N. Zhou, "Estimating total energy consumption and emissions of China's commercial and office buildings," 2008.
- [6] Y. Wu, "China building energy efficiency: current status, issues, and policy recommendations," *China Ministry of Construction*, 2003.
- [7] N. Zhou, "Energy use in commercial building in China: Current situation and future scenarios," *8th ECEEE Summer Study, Lawrence Berkeley National Laboratory*, pp. 1065-1071, 2007.
- [8] "Singapore Energy Statistics," 2016.
- [9] K. Chua, S. Chou, W. Yang, and J. Yan, "Achieving better energy-efficient air conditioning—a review of technologies and strategies," *Applied Energy*, vol. 104, pp. 87-104, 2013.
- [10] "<http://www.betterbuildingspartnership.com.au/information/fan-coil-unit-systems/>," *Better Building Partnership*.
- [11] C. Stetiu, "Energy and peak power savings potential of radiant cooling systems in US commercial buildings," *Energy and Buildings*, vol. 30, pp. 127-138, Jun 1999.
- [12] L. Z. Zhang and J. L. Niu, "Indoor humidity behaviors associated with decoupled cooling in hot and humid climates," *Building and Environment*, vol. 38, pp. 99-107, 2003.
- [13] J. Dieckmann and R. Zogg, "Chilled beam cooling," *Ashrae Journal*, vol. 49, 2007.

References

- [14] P. Rumsey and J. Weale, "Chilled beams in labs: eliminating reheat & saving energy on a budget," *Ashrae Journal*, vol. 49, 2007.
- [15] S. Weidner, J. Doerger, and M. Walsh, "Cooling with less air: using underfloor air distribution and chilled beams," *Ashrae Journal*, vol. 51, pp. 34-41, 2009.
- [16] M. H. J. D. A. Bouza, "Emerging Technologies: DOAS, Radiant Cooling Revisited," *Ashrae*, 2012.
- [17] P. Simmonds, "To beam or not to beam," *Engineered Systems*, 2013.
- [18] 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, 2014.
- [19] "Active and Passive Beam Application Design Guide", *Joint publication of REHVA and ASHRAE*, 2015.
- [20] P. Rumsey, N. Bulger, J. Wenisch, and T. Disney, "Chilled Beams in Laboratories: Key strategies to ensure effective design, construction, and operation," *Labs21* http://www.i2sl.org/documents/toolkit/bp_chilled-beam_508.pdf, 2009.
- [21] K. M. P. P. E. J. Leffingwell, "Chilled beams: The new system of choice?," *Ashrae*, vol. *Wisconsin Chapter*.
- [22] C. Lowell, "Don't turn active beams into expensive diffusers," *Ashrae Journal*, vol. 54, 2012.
- [23] M. Sandberg, B. Wiren, and L. Claesson, "Attachment of a cold jet to the ceiling—length of recirculation region and separation distance," *Proceedings of Roomvent 92, Aalborg, Denmark*, vol. 499, 1992.
- [24] H. B. Awbi and A. Hatton, "Mixed convection from heated room surfaces," *Energy and buildings*, vol. 32, pp. 153-166, 2000.
- [25] O. Seppanen, W. J. Fisk, and M. J. Mendell, "Ventilation rates and health," 2002.
- [26] O. A. Seppanen and W. J. Fisk, "Summary of human responses to ventilation," 2004.
- [27] Z. Guan and C. Wen, "Numerical investigation of geometry parameters for designing efficient terminal units in active chilled beam," in *Industrial Electronics and Applications (ICIEA)*, pp. 1114-1118, 2014.

References

- [28] Z. Guan and C. Wen, "Geometric optimization on active chilled beam terminal unit to achieve high entrainment efficiency," *Applied Thermal Engineering*, vol. 98, pp. 816-826, 2016.
- [29] C. Chen, W. Cai, Y. Wang, and C. Lin, "Further study on the heat exchanger circuitry arrangement for an active chilled beam terminal unit," *Energy and Buildings*, vol. 103, pp. 352-364, 2015.
- [30] C. Chen, W. Cai, K. Giridharan, and Y. Wang, "A hybrid dynamic modeling of active chilled beam terminal unit," *Applied Energy*, vol. 128, pp. 133-143, 2014.
- [31] "ASHRAE Standard 200-2015: Methods of Testing Chilled Beams", *ASHRAE Standard*, 2015.
- [32] P. Filipsson, A. Trüschel, J. Gräslund, and J.-O. Dalenbäck, "A thermal model of an active chilled beam," *Energy and Buildings*, vol. 149, pp. 83-90, 2017.
- [33] C. L. Cammarata G. Iacono M., Petrone G.*, "A CFD study on active chilled beams for indoor air conditioning," *the Proceedings of the COMSOL Users Conference*, 2007.
- [34] "BS EN 15116: 2008 "Ventilation in buildings-Chilled beams-Testing and rating of active chilled beams", " *British Standards Institution*, 2008.
- [35] A. Maccarini, G. Hultmark, A. Vorre, A. Afshari, and N. C. Bergsøe, "Modeling of active beam units with Modelica," in *Building Simulation*, pp. 543-550, 2015.
- [36] D. CLERCQ B, D. B, and V. O. J, "Measuring and modelling heat exchange capacity of active chilled beams," *11th REHVA World Congress and 8th International Conference on Indoor Air Quality, Ventilation and Energy Conservation in Buildings*, 2013.
- [37] "ASHRAE Standard Fundamentals," *American Society of Heating, Refrigerating and Air Conditioning Engineers, Atlanta*, vol. 111, 2001.
- [38] K. Hagström, E. Sandberg, H. Koskela, and T. Hautalampi, "Classification for the room air conditioning strategies," *Building and environment*, vol. 35, pp. 699-707, 2000.
- [39] C. Lubert, "On some recent applications of the Coanda effect to acoustics," *Proceedings of Meetings on Acoustics*, 2010

References

- [40] D. I. H. Jackmann, "How the air throw of air coolers is influenced by the spatial geometry and by air cooling," *Guntner AG & Co. KG*, 2008.
- [41] R. Neuendorf and I. Wygnanski, "On a turbulent wall jet flowing over a circular cylinder," *Journal of Fluid Mechanics*, vol. 381, pp. 1-25, 1999.
- [42] B. Wu, W. Cai, H. Chen, and K. Ji, "Experimental investigation on airflow pattern for active chilled beam system," *Energy and Buildings*, vol. 166, pp. 438-449, 2018.
- [43] A. Verhoff, "The two-dimensional, turbulent wall jet with and without an external free stream," *DTIC Document*, 1963.
- [44] W. Schwarz and W. Cosart, "The two-dimensional turbulent wall-jet," *Journal of Fluid Mechanics*, vol. 10, pp. 481-495, 1961.
- [45] 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, 2000.
- [46] G. Cao, M. Sivukari, J. Kurnitski, and M. Ruponen, "PIV measurement of the attached plane jet velocity field at a high turbulence intensity level in a room," *International Journal of Heat and Fluid Flow*, vol. 31, pp. 897-908, 2010.
- [47] G. Cao, C. Kandzia, D. Müller, J. Heikkinen, R. Kosonen, and M. Ruponen, "Experimental study of the effect of turbulence intensities on the maximum velocity decay of an attached plane jet," *Energy and Buildings*, vol. 65, pp. 127-136, 2013.
- [48] H. B. Awbi, *Ventilation of buildings*, 2002.
- [49] G. Cao, "Modelling the attached plane jet in a room," 2009.
- [50] V. Zbořil, A. Melikov, B. Yordanova, L. Bozhkov, and R. Kosonen, "Airflow distribution in rooms with active chilled beams," in *Proceedings of Roomvent*, 2007.
- [51] H. Koskela, H. Hägglom, 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, 2010.
- [52] H. Koskela, H. Hägglom, 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.

References

- [53] R. Kosonen, A. Melikov, L. Bozhkov, and B. Yordanova, "Impact of heat load distribution and strength on airflow pattern in rooms with exposed chilled beams," in *Proceedings of Roomvent*, 2007.
- [54] 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, 2001.
- [55] 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, 2010.
- [56] "ASHRAE Standard 62.1 - 2010 Ventilation for Acceptable Indoor Air Quality," *ASHRAE Standard*, 2010.
- [57] "ASHRAE Standard 113-2005 Method of testing for room air diffusion," *ASHRAE Standard*, 2005.
- [58] R. F. Rupp, N. G. Vásquez, and R. Lamberts, "A review of human thermal comfort in the built environment," *Energy and Buildings*, vol. 105, pp. 178-205, 2015.
- [59] P. O. Fanger, "Thermal comfort. Analysis and applications in environmental engineering," *Mc. Graww Hill*, 1970.
- [60] R. De Dear and G. S. Brager, "Developing an adaptive model of thermal comfort and preference," *CRC press*, 1998.
- [61] J. True, V. Zboril, R. Kosonen, and A. Melikov, "Consideration for minimising draught discomfort in Rooms with Active Chilled Beams," *Proceedings of Clima, Wellbeing Indoors*, 2007.
- [62] R. Kosonen, P. Mustakallio, A. Melikov, and M. Duszyk, "Comparison of the Thermal Environment in Rooms with Chilled Beam and Radiant Panel Systems," in *Proceedings of Roomvent conference. Trondheim*, 2011.
- [63] R. Kosonen, M. Virta, and A. Melikov, "The impact of thermal loads on indoor air flow," in *Proceedings of CLIMA*, 2007.
- [64] "Standard 55-2010 Thermal environmental conditions for human occupancy," *ASHRAE Standard*, 2010.

References

- [65] E. Iso, "7730-2005: "Ergonomics of the thermal environment—Analytical determination and interpretation of thermal comfort using calculation of the PMV and PPD indices and local thermal comfort criteria", " *International Organization for Standardisation, Geneva*, 2005.
- [66] K.-N. Rhee, M.-S. Shin, and S.-H. Choi, "Thermal uniformity in an open plan room with an active chilled beam system and conventional air distribution systems," *Energy and Buildings*, vol. 93, pp. 236-248, 2015.
- [67] H. S. L. C. Hens, "Thermal comfort in office buildings: Two case studies commented," *Building and Environment*, vol. 44, pp. 1399-1408, 2009.
- [68] B. Wu, W. Cai, and K. Ji, "Heat source effects on thermal comfort for active chilled beam systems," *Building and Environment*, vol. 141, pp. 91-102, 2018.
- [69] K. Chung and C. Lee, "Predicting air flow and thermal comfort in an indoor environment under different air diffusion models," *Building and Environment*, vol. 31, pp. 21-26, 1996.
- [70] M. Frontczak and P. Wargocki, "Literature survey on how different factors influence human comfort in indoor environments," *Building and Environment*, vol. 46, pp. 922-937, 2011.
- [71] T. Yao and Z. Lin, "An experimental and numerical study on the effect of air terminal types on the performance of stratum ventilation," *Building and Environment*, vol. 82, pp. 431-441, 2014.
- [72] T. Yao and Z. Lin, "An experimental and numerical study on the effect of air terminal layout on the performance of stratum ventilation," *Building and Environment*, vol. 82, pp. 75-86, 2014.
- [73] A. Melikov, B. Yordanova, L. Bozhkov, V. Zboril, and R. Kosonen, "Human response to thermal environment in rooms with chilled beams," in *Proceedings of CLIMA*, 2007.
- [74] A. K. Melikov, B. Yordanova, L. Bozhkov, V. Zboril, and R. Kosonen, "Impact of the airflow interaction on occupants' thermal comfort in rooms with active chilled beams," in *The 6th International Conference on Indoor Air Quality, Ventilation and Energy Conservation in Buildings*, 2007.

References

- [75] P. Fanger and C. Pedersen, "Discomfort due to air velocities in spaces," in *Proceedings of the Meeting of Commission B*, 1977.
- [76] T. L. Madsen, *Limits for Draught and Asymmetric Radiation in relation to human thermal well-being*: Thermal Insulation Laboratory, Technical University of Denmark, 1978.
- [77] H. Hensel and H. Hensel, *Thermoreception and temperature regulation*, 1981.
- [78] E. Mayer, "Physical causes for draft: some new findings," *Ashrae Transactions*, vol. 93, pp. 540-548, 1987.
- [79] A. K. Melikov, H. Hanzawa, and P. O. Fanger, "Airflow characteristics in the occupied zone of heated spaces without mechanical ventilation," *Ashrae Transactions*, 1988.
- [80] P. O. Fanger, A. K. Melikov, H. Hanzawa, and J. Ring, "Air turbulence and sensation of draught," *Energy and Buildings*, vol. 12, pp. 21-39, 1988.
- [81] H. Awbi, "Application of computational fluid dynamics in room ventilation," *Building and Environment*, vol. 24, pp. 73-84, 1989.
- [82] W. Chow, "Application of computational fluid dynamics in building services engineering," *Building and Environment*, vol. 31, pp. 425-436, 1996.
- [83] P. J. Jones and G. E. Whittle, "Computational fluid dynamics for building air flow prediction—current status and capabilities," *Building and Environment*, vol. 27, pp. 321-338, 1992.
- [84] P. V. Nielsen, "Computational fluid dynamics and room air movement," *Indoor Air*, vol. 14, pp. 134-143, 2004.
- [85] Z. Zhai, "Application of computational fluid dynamics in building design: aspects and trends," *Indoor and Built Environment*, vol. 15, pp. 305-313, 2006.
- [86] 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, 2014.
- [87] 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, 2003.

References

- [88] P. V. Nielsen, "Fifty years of CFD for room air distribution," *Building and Environment*, vol. 91, pp. 78-90, 2015.
- [89] G. Einberg, K. Hagström, P. Mustakallio, H. Koskela, and S. Holmberg, "CFD modelling of an industrial air diffuser—predicting velocity and temperature in the near zone," *Building and Environment*, vol. 40, pp. 601-615, 2005.
- [90] 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. 765-770.
- [91] H. Koskela, "Momentum source model for CFD-simulation of nozzle duct air diffuser," *Energy and Buildings*, vol. 36, pp. 1011-1020, 2004.
- [92] B. Wu, W. Cai, Q. Wang, C. Chen, C. Lin, and H. Chen, "The Air Distribution Around Nozzles Based On Active Chilled Beam System," *International High Performance Buildings Conference*, 2016.
- [93] J. Woollett and J. Rimmer, "Active and Passive Beam Application Design Guide," *Brussels: REHVA-Federation of European Heating, Ventilation and Air Conditioning Associations*, 2014.
- [94] F. N. Fritsch and R. E. Carlson, "Monotone piecewise cubic interpolation," *SIAM Journal on Numerical Analysis*, vol. 17, pp. 238-246, 1980.
- [95] R. Kosonen, P. Saarinen, H. Koskela, and A. Hole, "Impact of heat load location and strength on air flow pattern with a passive chilled beam system," *Energy and Buildings*, vol. 42, pp. 34-42, 2010.
- [96] I. C. Nelson, C. H. Culp, J. Rimmer, and B. Tully, "The effect of thermal load configuration on the performance of passive chilled beams," *Building and Environment*, vol. 96, pp. 188-197, 2016.
- [97] S. Lestinen, S. Kilpeläinen, R. Kosonen, J. Jokisalo, and H. Koskela, "Experimental study on airflow characteristics with asymmetrical heat load distribution and low-momentum diffuse ceiling ventilation," *Building and Environment*, 2018.

References

- [98] M. Vellei, M. Herrera, D. Fosas, and S. Natarajan, "The influence of relative humidity on adaptive thermal comfort," *Building and Environment*, vol. 124, pp. 171-185, 2017.
- [99] S. Jing, B. Li, M. Tan, and H. Liu, "Impact of relative humidity on thermal comfort in a warm environment," *Indoor and Built Environment*, vol. 22, pp. 598-607, 2013.
- [100] M. Virta, D. Butler, J. Graslund, J. Hogeling, and E. Kristiansen, "Chilled Beam Application Guidebook, Rehva Guidebook No 5," 2005.
- [101] V. Oyj, "HUMIDITY CONVERSION FORMULAS-Calculation formulas for humidity," *Humidity Conversion Formulas*, p. 16, 2013.
- [102] M. S. Tremblay, S. Aubert, J. D. Barnes, T. J. Saunders, V. Carson, A. E. Latimer-Cheung, *et al.*, "Sedentary Behavior Research Network (SBRN)–Terminology Consensus Project process and outcome," *International Journal of Behavioral Nutrition and Physical Activity*, vol. 14, 2017.
- [103] M. o. t. E. Institute of Environmental Epidemiology, Singapore, "Guidelines for good indoor air quality in office premises," 1996.
- [104] X. Wang, W. Cai, J. Lu, Y. Sun, and L. Zhao, "Model-based optimization strategy of chiller driven liquid desiccant dehumidifier with genetic algorithm," *Energy*, vol. 82, pp. 939-948, 2015.
- [105] L. Lu, W. J. Cai, Y. S. Chai, and L. H. Xie, "Global optimization for overall HVAC systems - Part I problem formulation and analysis," *Energy Conversion and Management*, vol. 46, pp. 999-1014, 2005.