# DOCTOR THESIS

**JUNE 2021**

# Study on Indoor Environment and Indoor Humidity Distribution using Computational Fluid Dynamics

The University of Kitakyushu
Faculty of Environmental Engineering

Department of Architecture

Ryu Laboratory

**Zhang Fangyuan**

## THE MAIN SYMBOL TABLE

### MAIN SYMBOL

**a** ------------------ Accessibility, Dimensionless

**Ar** -------------------Archimedes number, Dimensionless

**C** --------------------Concentration, $g/m^3$

**$D_{AB}$** --------------------------Mass diffusion coefficient, $m^2/s$

**d** --------------------Absolute humidity, g/kg

**J** --------------------Distribution, g/s

**i, j** -------------------Common symbol, Dimensionless

**N** -------------------The total amount , Dimensionless

**n** ------------------- Count, Dimensionless

**Q** ------------------ Room ventilation mass flow, kg/s

**T** -------------------Temperature, ℃

**U** -------------------Velocity, m/s

**V** -------------------Room volume, $m^3$

### GREEK ALPHABET

**α** -------------------Mass flow transfer coefficient, m/s

**$\Gamma_{Ceff}$** --------------------Equivalent diffusion coefficient, kg/(m*s)

**$\Gamma_{eff}$** --------------------Equivalent heat diffusion coefficient, kg/(m*s)

**τ** ------------------Time, s

**ρ** ------------------Density, $kg/m^3$

**σ** ------------------Mass transfer schmidt number, Dimensionless

**$\sigma_t$** ----------------- Turbulence mass transfer schmidt number, Dimensionless

**ν** ------------------Kinematic viscosity, $m^2/s$

**μ** ------------------Laminar dynamic viscosity, kg/ (m*s)

**$\mu_t$** ------------------Turbulence viscosity coefficient, kg/ (m*s)

### SUPERSCRIPT

**N** ------------------The total amount

**n** -------------------Count

**P** -------------------A point in the room

**SUBSCRIPT**

**0**----------------------Initial state

**b** ---------------------Boundary

**C** ---------------------Source

**R** ---------------------Return air

**r** ----------------------Room

**S** ---------------------Supply air

**I** ----------------------Initial distribution

**ABBREVIATIONS**

**AHU** -----------------------Air Handling Unit

**CFD** -----------------------Computational Fluid Dynamics

**HVAC** ---------------------Heating, Ventilation and Air Conditioning

**TASA** --------------------- Transient Accessibility of Supply Air

**TAIC** -----------------------Transient Accessibility of Initial Condition

**TAWS** ----------------------Transient Accessibility of Wet Source

# Abstract

According to statistics, people will spend 80% of their lives indoors. With the development of air conditioning technology, research into the indoor air environment has shifted from comfort to healthy air conditioning. Effective ventilation and reasonable airflow organization are important for improving indoor air quality. Humidity is an important parameter for evaluating indoor air quality, not only affecting the thermal comfort of humans, but also seriously limiting the performance of buildings.

The main methods used to study the indoor environment from a ventilation and air conditioning perspective are model experiments and computational fluid dynamics methods. CFD is increasingly favored for its unique advantages of low cost, speed, complete information and the ability to simulate a variety of different operating conditions. Airpak is a CFD system analysis software that simulates the temperature, humidity, velocity, air mean age, pollutant concentration, PMV and PPD fields of a room under different airflow organization of the air conditioning system to make a comprehensive evaluation of the airflow organization, thermal comfort and indoor air quality (IAQ).

This paper uses CFD technology and the Airpak software to numerically simulate the indoor microclimate environment. On the basis of the simulation results, the thermal comfort indexes of the human body are compared and evaluated.

In order to test the feasible of CFD method, the simulation result was analyzed and checked by experiment data, and the results are validated by actual experiment. It is hoped that the research results can control of the decision-making of indoor temperature and humidity distribution.

After induction, in chapter 2 this paper analyses the factors affecting indoor humidity distribution from a mathematical theoretical point of view, describes the principles and methods of calculation based on the CFD method and on this basis, determines the feasibility of the technique for indoor air temperature and humidity simulation. CFD simulations of indoor temperature and humidity in the laboratory were carried out and field measurements were taken. The simulation results were compared with the experimental measurements. The results show that the CFD simulation results are in good agreement with the actual test results. The calculation of indoor air ambient temperature and humidity is feasible and accurate.

In Chapter 3, a simple physical model was developed to study the influence of different ventilation methods on the humidity distribution and condensation formation conditions in the room. The simulation results show that the ventilation method has a significant influence on the condensation time, the amount of condensation and the distribution pattern of condensation on the indoor wall surfaces. The best way to control indoor air humidity is to use the bottom-in, top-out method with supply and return air outlets on both sides of the room, which can significantly reduce the occurrence of condensation. The analysis of the indoor air mean age also

demonstrates the higher indoor air quality with the bottom air supply method.

Based on the simulation results in Chapter 4, a small bachelor flat was used to study the effect of the placement and height of the humidifier and the arrangement of the room supply and return air outlets on the indoor humidity distribution in winter. And it was obtained that by placing the humidifier at a height of 0.5 m, the average humidity in the room was higher than if it was placed on the floor. Placing the humidifier at the entrance of the room did not significantly improve the overall humidity in the room. It is also important to avoid placing humidifiers near windows to avoid condensation.

Currently, the common central air conditioning systems used in offices are all-air primary return air system and VRV multi-connector plus fresh air air-conditioning system. In chapter 5 ,Airpak-based VRV plus fresh air air-conditioning system model and primary return all air air-conditioning system model were established to simulate the temperature field and velocity field distribution of indoor airflow under the two working conditions in winter, and PMV-PPD and air age indexes were used to evaluate the thermal comfort of indoor personnel and air quality. The results show that the air velocity in the working area of the two solutions can meet the requirements of comfort air-conditioning, personnel do not have obvious blowing sensation, primary return air air-conditioning system compared with VRV plus fresh air air-conditioning system: airflow disturbance is smaller, temperature fluctuation in the breathing area of personnel is smaller, fresh air can be delivered in time, and indoor air quality is better.

# CHAPTER 1
# INTRODUCTION

# 1. Introduction

## 1.1. Background

Building energy consumption and indoor air quality are two important issues concerned in the field of built environment. With the increasing degree of global industrialization and the continuous improvement of people's living standards, more and more attention has been paid to indoor thermal comfort.

While the national economy is improving, it also brings tremendous pressure on the environment. The pollution of water environment and the deterioration of atmospheric environment, these practical environmental problems have brought great pressure to sustainable development, but also brought many problems to people's living environment. Indoor environment is an important place closely related to People's Daily work and life. According to statistics, people's life will be about eighty percent of the time spent indoors, the stand or fall of indoor air environment and people's health has a close relationship and directly, the quality of air environment not only affects people's psychology and behavior, the more influence on people's physical and health, because indoor air environment quality problems caused by the human body discomfort or even disease have become common, so indoor air quality is about the safety of the people, healthy and comfortable (Fanger,2004). And according to the World Health Organization (WHO) and scholars around the World. Nearly two hundred and thirty years to the indoor air quality problem is increasingly serious in many countries, life and work for a long time in bad people show more and more serious in the indoor environment of pathological reaction, this issue has caused wide attention of experts and scholars, and put forward the sick building soon (sick building) and sick building syndrome (SBS, sick building syndrome) and BRI (building related illness). According to the definition of the World Health Organization (WHO) in 1983, sick building syndrome is a condition caused by the use of buildings, including redness of the eyes, runny nose, sore throat, headache, dizziness, nausea and itchy skin (World Health Organization, 1983; World Health Organization, 1989; Meininghaus et al., 1999; Wang et al., 1998). Experts analyzed that most of these symptoms are caused by the high concentration of volatile organic compounds (VOCs) in the indoor air environment. In order to realize the building ecology, it is necessary to provide people with a safe, healthy and comfortable indoor air environment with high efficiency. For this reason, the spatial distribution characteristics of VOCs concentration in indoor air environment have become an important topic in modern indoor environment research.

Healthy and moist skin should be the state that water is locked by corneous layer on the surface of the skin. The importance of the skin is the interface for the body and the external environment, and is the first line of defense foreign influence, such as the skin in protecting the body from pathogens. And to avoid excessive water loss has important role. Skin has epidermis, dermis and subcutaneous tissue three-layer structure and sweat glands, sebaceous glands and hair accessory organs. The surface of the skin cutin layer with a layer of the skin the sebaceous glands secretion of sebum, cutin cell lipid and out of the sweat glands secretion of sweat and dust in the air, bacteria, germs, and so on, formed by the fusion of a layer of film is called sebum membrane. Sebum membrane is the most important of a layer of skin moisture, effectively locks in moisture, prevent skin moisture to evaporate transition, and can prevent external moisture and some material penetration in great quantities, as a result, the skin's moisture content to keep normal state. Stay in dry and cold climate in winter, because of the sweat and sebum secretion reduced, often in a state of incomplete cortex membrane, moisture to evaporate from the gap. So for a long time in a dry environment, the surface of the skin cut in layer will be because of the dry desquamation, and cracks. The amount of sebum coming discharged from the sebaceous glands is regulated by hormones. Most people know the effect of indoor room temperature condition on the health, but the indoor humidity conditions are not being taken seriously. However, the great influence of humidity cannot be ignored. Humidity is the amount of water vapor in the air. Water vapor is the gaseous state of water and is invisible. Absolute humidity is the total mass of water vapor present in a given volume of air. It does not take temperature into consideration. Absolute humidity in the atmosphere ranges from near zero to roughly 30 grams per cubic meter when the air is saturated at 30 ℃. The absolute humidity changes as air temperature or pressure changes. The relative humidity of an air-water mixture is defined as the ratio of the partial pressure of water vapor in the mixture to the equilibrium vapor pressure of water at a given temperature.

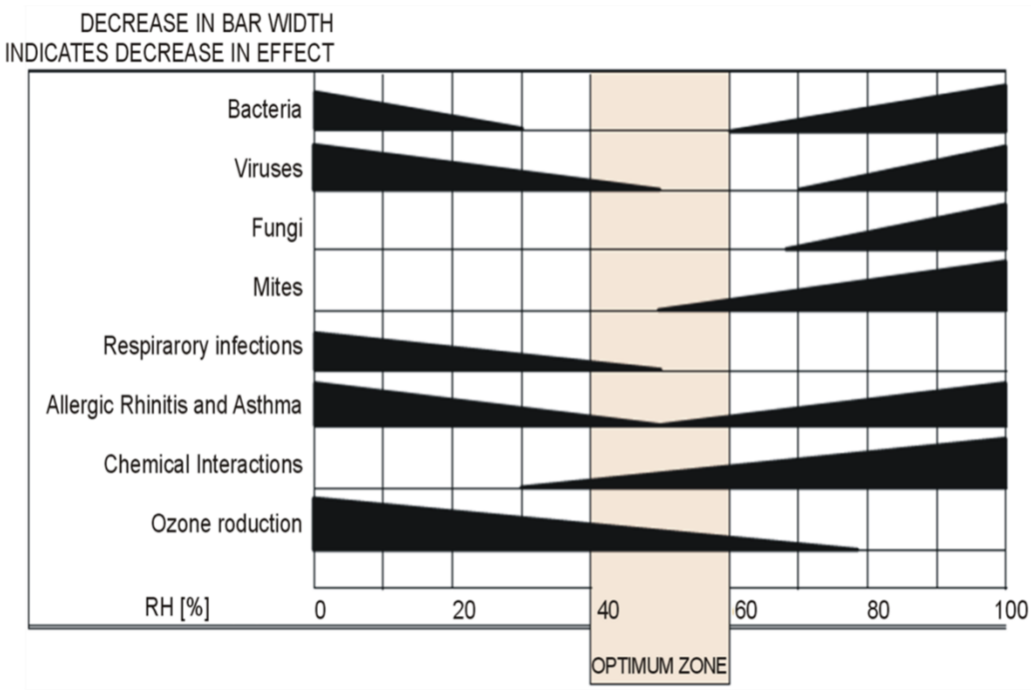



Figure 1-1 Optimum zone of relative humidity

*Ventilation and Relative Humidity in Swedish Buildings,January 2014Journal of Environmental Protection 05(11):1022-1036,DOI:10.4236/jep.2014.511102;Authors:Thomas AlsmoCatharina Alsmo

The optimum relative humidity for the human body is 40 to 60 percent(Fig1-1). When the indoor humidity less than 40 percent, throat and mucous membranes inside the nasal cavity will become dry, it will cause the inflammation. This will not only make people feel uncomfortable, and may lead to human immunity of bacteria and viruses decreased. Further, ASHER Canadian study shows that at 40-60% relative humidity, the growth of bacteria, fungi, viruses and microorganisms is inhibited Not only that, respiratory disease and allergic rhinitis, also symptoms such as asthma, it has been found the most unlikely thing at the time of the relative humidity of 40 - 60%. A direct impact on the body is air-dried for dry skin. When the skin is dry, the skin allergy symptoms become worse.

At present, New Coronary Pneumonia -19 erupts in the world, and indoor environment has become the focus of everyone's attention. The ability of the virus to survive in the air depends on environmental conditions, especially ambient temperature and relative humidity are the most important parameters. A series of studies have shown that the dry environment seems to create an excellent environment for the growth and spread of influenza viruses. It is imperative to study the indoor humidity distribution Meteorological parameters are the important factors influencing the infectious diseases such as severe acute respiratory syndrome (SARS) and influenza.

In December 2019, a novel coronavirus disease (COVID-19) epidemic was reported in Wuhan, China, which is caused by severe acute respiratory syndrome coronavirus 2 (SARS-CoV-2) (Gorbalenya, 2020; Wu et al., 2020). The COVID-19 has been affirmed to have human-to-human transmissibility (C. Wang et al., 2020; M. Wang et al., 2020), which raised high attention not only in China but internationally. The World Health Organization (WHO) reported that there are 118,319 confirmed cases and 4292 deaths globally until March 11, 2020 (WHO, 2020a), and evaluated as global pandemic on the same day (WHO, 2020b).

In retrospect studies, the outbreak of severe acute respiratory syndrome (SARS) in Guangdong in 2003 gradually faded with the warming weather coming, and was basically ended until July (Wallis and Nerlich, 2005). It has been documented that the temperature and its variations might have affected the SARS outbreak (Tan, 2005). A study in Korea found that the risk of influenza incidence was significantly increased with low daily temperature and low relative humidity, a positive significant association was observed for diurnal temperature range (DTR) (Park et al., 2019). Moreover, temperature (Pinheiro et al., 2014) and DTR (Luo et al., 2013) have been linked to the death from respiratory diseases. A study demonstrated that absolute humidity had significant correlations with influenza viral survival and transmission rates (Metz and Finn, 2015). Few studies reported that the COVID-19 was related to the meteorological factors, which decreased with the temperature increasing (Oliveiros et al., 2020; C. Wang et al., 2020; M. Wang et al., 2020), but their effects on the mortality have not been reported.

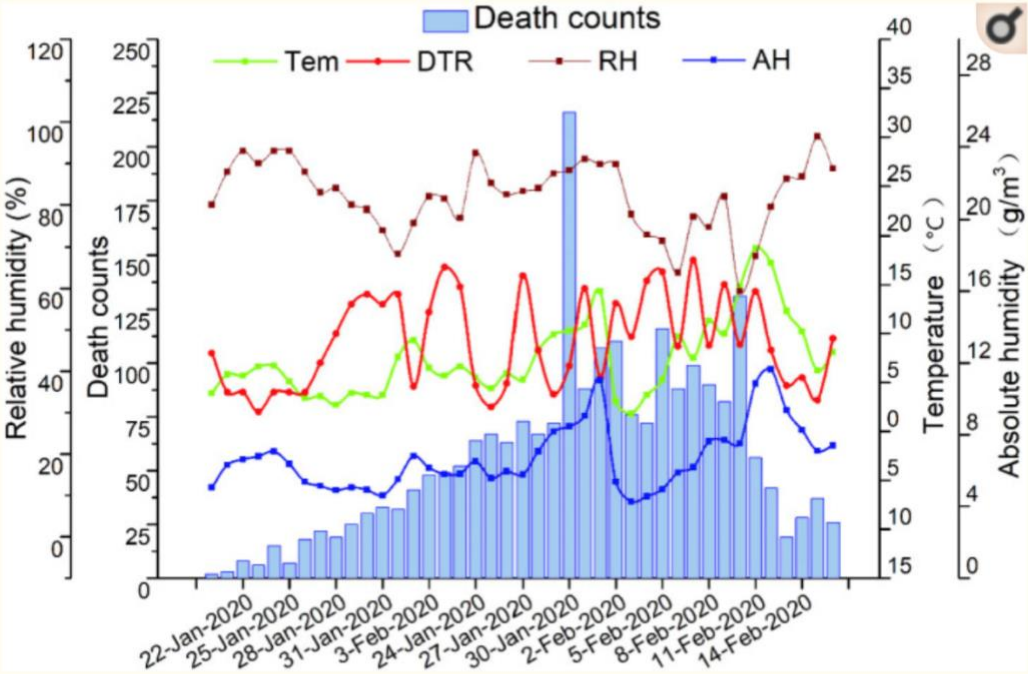



Fig. 1

Temporal pattern of COVID-19 daily mortality and meteorological factor levels in Wuhan, China, from 20 January to 29 February 2020. COVID-19, Corona Virus Disease 2019; Tem, temperature; DTR, diurnal temperature range; RH, relative humidity; AH, absolute humidity.

Figure 1-2 COVID-19 and Humidity

*（ Effects of temperature variation and humidity on the death of COVID-19 in Wuhan, China Yueling Ma,a,1 Yadong Zhao,b,1 Jiangtao Liu,a Xiaotao He,a Bo Wang,a Shihua Fu,a Jun Yan,c Jingping Niu,a Ji Zhou,d,e and Bin Luoa,e,f,⁎ Sci Total Environ. 2020 Jul 1; 724: 138226. ）

According to research background, indoor environment is an important place that closely related people's daily work and life. Humidity is an important indicator of indoor air parameters. And Japan is accelerating into an aging society. More and more elderly start using welfare facilities. In winter time, heater is used frequently. The influence of ventilation and humidify can not be ignored. If the winter indoor air is too dry, it will be a great influence to the health of living, especially for the elderly. In order to improve the quality of life in the elderly, the improvement of the living environment is indispensable. Most of the elderly can know the indoor temperature regime through thermometer, but they always ignore the indoor humidity regime. The elderly are insufficient understanding of the importance of the indoor humidity. In addition, Indoor humidity conditions are strongly associated with the survival rate of virus. So the humidity control is particularly important.

We also found that when using the humidifier there are some parts still dry in the room. Indoor humidity distribution is not uniform, when using humidifier. So the humidifier reasonable placement and effective humidification range need to be researched.

In addition, the room walls, roofs and ground condensation are also a serious problem. Wall condensation is due to the moisture content of the air near the wall is greater than the wall temperature of the saturated moisture content. Wall condensation will cause structural corrosion, moldy mold, indoor air quality decline and other problem.

Due to the use of lumped parameters of the method can not solve the non-uniform environment such as condensation. Combined with the existing research results, the study of indoor water vapor transmission and air humidity distribution has become the key to solve humidity problems.

Therefore, how to create a building indoor environment that is healthy, comfortable, and safe without causing a significant increase in operating energy consumption is an important strategic issue that China must face and solve, as well as the world's HVAC field. Major technical issues. The composition of HVAC system mainly includes three parts: cold and heat source, transmission and distribution system, end mode and equipment. The research on cold and heat sources and transmission and distribution systems is relatively mature, and with the in-depth application of various new and improved cold and heat source forms and optimized transmission and distribution methods in practical engineering, the energy saving potential of cold and heat sources and transmission and distribution systems is continuously Excavation has now tended to improve. The end form (including the airflow organization and equipment form) not only directly determines the final indoor air environment, but also plays an important role in the creation of the indoor air environment. Compared with the cold and heat sources and transmission and distribution systems, the end of the indoor environment has Greater energy saving potential. The direct function of the end of the indoor environment is to produce environmental parameters such as temperature, humidity, and pollutant concentration that meet the needs. The safe, comfortable, and energy-saving indoor air environment is also determined by these parameters. Traditional building ventilation and air conditioning description of indoor temperature, humidity, pollutant concentration and other parameters usually uniforms the indoor environment and uses a lumped parameter approach (Annex, 1997). But for the actual ventilation, such as replacement air supply (Mundt, 1996; Chen et al, 2003), personalized air supply (Cermaket al, 2002; Melikov et al, 2002), and even mixed air supply (Heiselberg, 1996) The parameters of the ventilated room are uneven. With the deepening understanding of energy saving, in order to meet the human body's demand for air quality, it is not energy efficient and inefficient to ventilate indoors as uniform parameters (Fanger, 2001; Sun et al, 2008). Control has also become a trend to create an efficient and energy-saving indoor air environment (Cai et al, 2008), which plays a very important role in saving energy and improving system efficiency of ventilation and air conditioning systems.

The above background shows that reducing energy consumption and improving air quality are two key issues that need to be solved in the field of HVAC, and the importance of creating an indoor environment at this stage is particularly prominent. Therefore, how to create an indoor air environment that meets the needs and consumes less energy according to process requirements or human needs is the most important issue in the creation of indoor air environments. Since the traditional HVAC is mainly dedicated to creating a uniform indoor air environment, it is not only difficult to improve the ventilation efficiency and air quality, but also usually consumes more energy. If we can make a breakthrough in the theory of creating a non-uniform indoor air environment, it will definitely greatly promote the development of building energy conservation undertakings throughout the world. At the same time, it can be better applied to improving indoor air quality and other directions, and comprehensively promote HVAC Improvement of technology. Therefore, studying the creation method of non-uniform indoor air environment has not only important academic significance, but also very important application value.

## 1.2. Research fields and methods of indoor air environment

At present, there are four research methods for HVAC project: jet formula, Zonal model, CFD and model experiment.

(1) jet formula method

The jet formula method is based on the empirical formula to predict the specific form of airflow organization. Since the 1940s, many researchers have experimented and studied the jet characteristics of the ventilator in a ventilated room. And in the early fifties the establishment of a series of jet formula for indoor air distribution prediction, as the economy, a simple method of indoor air distribution prediction.

(2) regional model method (zonal model)

The 1970 Zonal Model method was proposed, and applied to the natural ventilation of the ventilation, temperature distribution and other aspects of the forecast calculation. The basic idea of the regional model method is to divide the room into a limited number of macroscopic regions, and uniformize the temperature and concentration of each region. There is a heat mass exchange between regions, applying the mass and energy conservation equations over the entire computational area, combining the coupling between regional flow and pressure, so as to study the temperature distribution and flow in the room. In recent years, some scholars have used this method to improve the air and particle deposition of single chamber and multi-chamber by zonal model method and CFD method.

(3) model experimental method

In the study of the model of natural ventilation, according to the principle of similarity, the wind pressure coefficient can be determined by studying the pressure distribution and velocity distribution of the surface of the model building under the natural ventilation condition. The wind pressure distribution at the exit of the building was measured by the wind tunnel model, and the air flow in the room was simulated by CFD according to the measured wind pressure coefficient as the boundary condition. In the model building with equal proportion or reduced proportion The measurement of indoor air distribution is made by means of measurement. Compared with the empirical theory analysis method, the model experiment method is more reliable, but the time and money are more costly. In the series of research on a problem, it may be necessary to adjust the model condition repeatedly, and analyze the sensitivity of the parameter The model experiment has been limited.

(4) CFD method

CFD is short for Computational Fluid Dynamics, a method that USES computer technology to solve Fluid Dynamics problems. Analysis of systems containing physical phenomena such as fluid flow and heat conduction through computer numerical calculations and image reality. To put it simply, it is the technique of using computer to solve various conservation control partial differential equations of fluid flow. And its basic principle is to put the original in time domain and space domain quantities of the straight, such as velocity field and pressure field, with a series of a finite number of discrete point set instead of a variable's value, and through certain principles and methods to establish the relationship of these discrete points out variable algebra equations, and then to solve the algebraic equations for variable approximation. Computational fluid dynamics (CFD) is the product of the combination of modern fluid mechanics, numerical calculation methods and computer graphics. It is a frontier science with strong vitality. Its basic characteristics are numerical simulation and computer experiments. Starting from the basic laws of physics, it USES electronic computer as a tool and applies various discrete mathematical methods to carry out numerical simulation, computer experiment and analysis of various problems in convection physics, so as to solve various practical problems. To a large extent, it has replaced the expensive fluid dynamics experiment equipment, which has made a great impact in scientific research and engineering technology. CFD can be considered as a modern simulation technology. CFD is one of the most concerned research fields in the world at present. It is the core and important technology for the research of "three transfer" (heat transfer, mass transfer, momentum transfer), combustion, multiphase flow and chemical reaction. It is widely used in many engineering fields such as thermal power, aerospace, machinery, civil water conservancy, hvac and refrigeration industry. The study of indoor air environment is one of the important fields of CFD application. Computers were developed to solve numerical calculation problems, while CFD was developed with the advancement of computer technology and numerical calculation methods. In 1933, British scientist Thom first used a hand computer to complete the numerical calculation of an externally swept cylinder flow. CFD was generated, but the application of computers and numerical methods to solve the flow and heat transfer problems gradually became large-scale and produced beneficial results around the world, roughly beginning in the 1960s. The development of CFD can be roughly divided into three stages: the first stage is 1965-1974, and the initial stage is mainly to solve the basic theory of computational fluid dynamics and the reliability analysis of numerical calculation; The second stage is from 1975 to 1984. In order to enter the industrial application stage, the feasibility of practical engineering application is mainly solved. The third stage is from 1985 to now, which has entered a period of rapid development. The practical application value of CFD has been further recognized, and the application of CFD has become increasingly widespread. CFD has developed rapidly since the 1960s. Nowadays, theoretical research, experimental analysis and numerical simulation have become three basic methods to study the law of fluid motion, and fluid research is divided into three interrelated fields. Especially in the last ten years, the application of CFD in practical engineering has made greater development. Almost all the problems involving fluid movement, heat exchange, molecular transport and other phenomena can be analyzed and simulated by means of computational fluid dynamics. CFD numerical simulation has been widely used in aerospace, dynamic engineering, mechanics, physics, chemistry, architecture, water conservancy, ocean, atmosphere, environment and other scientific research and engineering fields. Typical applications and related engineering problems are:

1. Fluid flow in hydraulic machinery such as turbines, fans and pumps;
2. Design of aircraft, space shuttle and other aircraft;
3. Influence of automobile streamline shape on performance;
4. Calculation of flood and estuary tidal current;
5. Influence of wind load on stability and structural performance of high-rise buildings;

6. Room temperature and indoor air flow and environmental analysis;
7. Cooling of electronic components;
8. Performance analysis of heat exchanger and selection of heat exchanger sheet shape;
9. Diffusion of pollutants in rivers;
10. Pollution of street environment by automobile exhaust;
11. Migration of bacteria in food.

Table 1-1 The compare of research methods for HVAC project

| **Project** | **Method** | | | |
|---|---|---|---|---|
| | **jet formula** | **zonal model** | **CFD** | **model experiment** |
| **Room shape** | Simple | Complex | Unlimited | Unlimited |
| **Dependency of experimental paramemter** | Complete | Very | A little | None |
| **Cost** | Lowest | Lower | Expensive | Most expensive |
| **Period** | Shortest | Short | Long | Longest |
| **Completeness of result** | Simple | Simple | Most detailed | Detailed |
| **Reliability** | Bad | Bad | Good | Excellent |
| **Applicability** | Mechanical ventilation about jet | Mechanical and natural ventilation under controllod condition | Mechanical and natural ventilation | Mechanical and natural ventilation |

CFD is an acronym for English Computational Fluid Dynamics. It is with the computer technology and numerical computing technology development and gradually developed. As early as 1933, the British Thom for the first time using a hand-cranked computer to successfully solve the two-dimensional viscous fluid partial differential equation, CFD was born. In short, CFD is equivalent to "virtual" in the computer to do experiments to simulate the actual simulation of fluid flow. The basic principle is to numerically solve the differential equation which controls the flow of fluid, and obtain the discrete distribution of the flow field of the fluid flow in the continuous region, which is similar to the simulation of the fluid flow, which is a kind of modern simulation

CFD has been developing rapidly in the last 30 years, in addition to the development of the computer hardware industry to provide a solid material basis, but also because the CFD method has the advantages of low cost and can simulate more complex or ideal process.

CFD is one of the most important research fields in the world. It is the core and important technology of heat transfer, mass transfer, momentum transfer and combustion, multiphase flow and chemical reaction. It is widely used in thermal energy, aerospace, Civil engineering, environmental chemical and many other engineering fields. HVAC refrigeration industry is one of the important areas of CFD technology applications.

In 1974, Nielsen first applied CFD technology to the field of HVAC engineering, numerical simulation of the air flow in the ventilation room. And after a short period of more than 30 years, CFD technology in HVAC engineering research and application has been widely promoted. Today, CFD technology has become a powerful HVAC engineers and architects to analyze and solve the problem of a powerful tool.

CFD is the technique of using computer to solve various kinds of conservation control partial differential equations of fluid flow, which involves hydrodynamics (especially turbulence mechanics), calculation method and even computer graphics processing technology. CFD technology will vary depending on the problem, such as the subsonic flow of compressible gases, low velocity flow of incompressible gases, and so on. For the flow of air conditioning in the field of flow problems, mostly low-speed flow, flow rate below 10m / s ; fluid temperature or density change is not, it can be seen as incompressible flow, do not have to consider the high flow of compressible fluid Wave and other complex phenomena.

Computational fluid dynamics is a new branch of numerical simulation and analysis of hydrodynamic problems using computer and discretized numerical methods. Although the history of computational fluid dynamics is not long, it has been extensively penetrated into various fields of fluid mechanics and quite a number of applications, and is still developing rapidly.

The CFD technology applied to HVAC projects has also been deepened and developed, and has been widely used in the following three types of HVAC projects:

1. Ventilation air conditioning airflow organization design

Ventilation and air conditioning space of the air flow form directly affect the ventilation and air conditioning effect, with CFD can predict the air distribution within the space, and then guide the air organization design. At the same time, it is also possible to simulate the temperature field, velocity field, humidity field and air quality in the ventilation and air conditioning space.

2. Outside the building environment analysis and design

The environment outside the building has an important influence on the life of the occupants in the building. The so-called secondary air and the thermal environment of the building are becoming more and more concerned. CFD can be used to simulate the external environment of the building, so as to design a reasonable building wind environment. Moreover, by simulating the wind and wind conditions outside the building, it can further guide the overall summer heat and winter insulation design, and even improve the natural ventilation within the building.

3. Research and improvement of construction equipment performance

HVAC works of many ancillary equipment, such as fans, processing units, air conditioning system, end devices, etc., are through the flow of fluid and the operation of the flow, so the flow of equipment performance has an important impact. Therefore, in order to improve the performance of the equipment, CFD simulation can be used to simulate the flow of its internal working fluid, and then improve it, to reduce building energy consumption and save operating costs. CFD is an analog simulation technology, in the field of HVAC engineering applications is mainly simulated space within the air or other fluid medium flow. The computational fluid dynamics method starts from the microscopic, divides the room into a large number of grids, introduces the governing equations of fluid mechanics, and discards the equations onto each grid. The finite difference method or the finite volume method, combined with the appropriate boundary conditions Solution, you can get the relevant physical quantities of each grid, such as speed, pressure and pollutant concentration. CFD method has the characteristics of high precision and strong applicability, and it can be widely used in the study of indoor air environment. Due to the diversity and complexity of the contents of indoor air environment research, in the process of research, planning and design of building ventilation and air conditioning, only by means of model experiment method and theoretical analysis (empirical and semi empirical) method to meet and reach the building The indoor air environment control requirements are difficult to achieve. The computational fluid dynamics (CFD) method, by virtue of its condition, has the advantages of arbitrariness and good reproducibility of the results, and gradually replaces the time-consuming, cost-effective model design and the traditional jet analysis method. In the design and research of ventilation and air conditioning More and more important role, researchers have also been in the indoor air environment research and engineering applications more and more use. The CFD method has become an important research tool for indoor air environment prediction analysis and testing of indoor air environment. On the other hand, CFD has the advantages of low cost, fast speed, complete data and can simulate various unique conditions, so it is gradually favored by people. Although CFD methods still have reliability and computability of practical problems, these problems have been developed and solved step by step. Therefore, the CFD method can be applied to the simulation and prediction of indoor air environment.

On the other hand, CFD has the advantages of low cost, fast speed, complete data and can simulate various unique conditions, so it is gradually favored by people. Although CFD methods still have reliability and computability of practical problems, these problems have been solved step by step. In order to study the indoor air distribution as an example, compared with other research methods, CFD method can quickly, accurately and low cost to get the room speed, temperature, humidity and harmful substances such as the physical distribution of the detailed distribution of the CFD method.

This paper using the CFD technique simulation of the humidity distribution of the laboratory. Compared with the actual measurement results, verify the feasibility of the CFD model. And in the CFD model to change the room air volume, exhaust volume and humidifier placement. Calculate the different air volume, exhaust volume and change the source location conditions, carried out the concentration and distribution of water vapor in air. Using the CFD method to study the humidity distribution in the indoor environment, and to provide reference for use of the indoor humidifier.

## 1.3. Research progress of CFD technology in indoor air environment

In 1974, Danish scholar P.V.Niels first used computational fluid dynamics, or CFD technology, to calculate indoor air flow. Since then, numerical simulation technology has been widely used in the field of HVAC engineering. At the beginning, due to the limitation of the computer's computing power, the problems that can be simulated have certain limitations. The main simulation is the indoor air flow in the two-dimensional model, and the indoor air velocity and temperature distribution are studied. From the late 1980s to the early 1990s, with the rapid improvement of computer capabilities and the development of numerical methods, Marakami and Chen and others began to use three-dimensional turbulence models to numerically simulate indoor ventilation problems, studying indoor air flow, ventilation effects, and Simulation of temperature and humidity coupling problems. Spitler.JD explores the movement of influent gases indoors when there is a temperature difference between indoor and outdoor. In the mid-to-late 1990s, researchers expanded the research object from a single indoor air flow to indoor pollutants and began using CFD technology to predict indoor air flow For the effect of pollutant diffusion, Fan used the k-ε model combined with the wall function method to numerically simulate indoor air flow and pollutant diffusion. Gan et al. Used the CFD calculation program VORTEX to numerically simulate the indoor air environment and analyzed the indoor PMV index and the distribution of pollutants. The research and application of CFD method in indoor air environment can be classified from three aspects: mechanism, method and engineering application.

(1) The research on the mechanism of indoor air environment includes the basic rules of indoor air velocity and temperature distribution, the effects of natural and mechanical ventilation on indoor environment, and the distribution of indoor pollutants (including gas phase and solid phase). In the mechanism research, the experimental method is more reliable and accurate. The CFD method as an auxiliary research section can replace the experimental technique to perform parameter sensitivity experiments under certain conditions,when the experimental conditions are difficult, numerical experiments can also be performed using the CFD method.

Chen conducted sensitivity studies on different boundary conditions of the air diffuser, and analyzed the influence of the calculated interpolation errors on the results.Awbi conducted a mechanism numerical study on the effect of the range of the Archimedes number and the Reynolds number on the velocity field and temperature field distribution. In the subsequent work, due to the convective heat transfer coefficient Comfort, air flow, and thermal loads all have an effect. Awbi uses the CM method to calculate the convective heat transfer coefficients of the heated floor, wall, and roof.

(2) The flow scale and characteristics of indoor air flow determine that most indoor air flows are turbulent flows. Due to the insufficient understanding of turbulence in fluid mechanics, the applicability and simplification of turbulence models has become a focus of method research.

Chen and Xu have proposed the zero-equation turbulence model and double-layer turbulence model for the study of indoor air environment. The new zero-equation turbulence model in the study is used to simulate natural convection, forced convection, mixed convection and indoor ventilation. Compared with the traditional k-ε model, this model saves more computing resources and speeds up more than 10 times. The double-layer turbulence model uses a single-equation turbulence model in the near-wall area and the k-ε model in the flow area away from the wall. It is in good agreement with the experimental results and saves computing resources. In 1987, Murakami applied large eddy simulation to the simulation of turbulence in a square cylinder. Kato et al. Conducted wind tunnel experiments and large eddy simulations on the velocity and pressure distribution of through-ventilation, and the results show that the large eddy simulations agree well with the experimental results. Bang and Chen performed numerical simulations of buoyancy-driven unilateral natural ventilation using the BANS model and catastrophe simulation, respectively, and compared with the results of wind tunnel experiments. The results show that the large eddy simulation performs better. In the field of mechanical ventilation, the use of large eddy simulation is relatively rare. Knowledge of the diffusion and deposition of particulate pollutants in the room can help to rationally design the indoor airflow organization. Zhao et al. Conducted a systematic study on the diffusion and deposition characteristics of particulate pollutants in an indoor environment. By comparing with measured data, they verified the performance of different Euler models and Lagrangian models in simulations. Studies show that Lagrang The daily DRW model and slip flux model obtained satisfactory prediction results. Li Xianting et al. Used computational fluid dynamics method instead of tracer gas to obtain the age distribution of air in the room through calculation. Zhao Bin et al. Used an error preprocessing method to iteratively solve the discrete algebraic equations for indoor air flow from coarse grids to fine grids. Compared with traditional methods, it significantly improved the convergence speed of numerical simulation. Yang Jianrong and others proposed indicators of pollutant accessibility to describe the non-uniformity of pollutant distribution in the new wind, and based on the actual system connection form, established a calculation model for the concentration distribution of pollutants in the return wind, which overcomes the traditional The inaccuracy of the lumped parameter method and the disadvantage of requiring multiple iterations.

(3) Applied research has always been carried out in the research work of indoor air environment. The CFD method is widely used in the study of the working efficiency of ventilation mode and the air environment assessment of buildings. Li Xianting and Sun Shufeng respectively studied the performance of the chair back air supply scheme and the dynamic air supply end device. It was found in the research that the positions of the air outlet and the air outlet can be adjusted to obtain better ventilation effect and environmental comfort, and it is more ideal to position the air supply device away from the human body area. The effects of the shape, position, air supply parameters, and placement of indoor furniture on the air environment in the workplace were compared through experiments and numerical simulations.

Based on the coupling characteristics of various influencing factors in the indoor air environment, when conducting a systematic study of the air environment, researchers need to couple the CFD method with other methods. Kim et al. Coupled the CFD method with radiant heat conduction processes and HVAC systems, and studied the indoor thermal environment. From the perspective of indoor organic pollutants, Yang et al. Constructed a numerical model combining indoor air flow with the throughput effect of VOCs.

## 1.4. Literature review

The air pollutant concentration, temperature, and humidity parameters in the indoor air environment are important parameters for the creation of the indoor air environment. The construction of the indoor environment includes design and operation. The key of the design link is how to accurately predict the indoor environment parameters; in the operation link, the key is how to control the indoor parameters to meet the design requirements of the indoor environment. For the design stage, it is necessary to have a deep understanding of the distribution law of indoor air parameters in order to grasp the law, so as to accurately describe and design prediction; In the operation stage, by optimizing the control method of the non-uniform environment, fast Efficiently and energy-efficiently control different environmental parameters at different locations indoors, thereby creating a reasonable indoor non-uniform environment.

This section will introduce the description and prediction methods of indoor non-uniform environment, and control methods of indoor non-uniform environment parameters. Because the numerical simulation method has the characteristics of short cycle, low cost, and can be performed in advance, this method has been greatly developed recently. At present, CFD technology has begun to be researched and applied in many industries and fields, mainly in the following aspects: ① Optimization and prediction of ventilation and air-conditioning design schemes, such as the numerical prediction (simulation) of air-conditioning design schemes of Changsha World Window Center Theater (Grand Theater), air space organization in high and large spaces, and replacement ventilation numerical simulation of the methods, numerical simulation of airflow distribution in clean rooms, etc .; ② CFD analysis of heat and mass transfer equipment, such as CFD analysis of various heat exchangers and cooling towers; ③ CFD analysis of jet technology, such as air conditioning air supply. CFD analysis of cold storage rooms and refrigeration equipment; ⑤ CFD analysis of fluid machinery and fluid components, such as pumps, fans and other rotating machinery, CFD analysis of various valves; ⑥ Air quality and thermal environment of the building Evaluation and prediction of CFD methods; ⑦CFD analysis of building fire smoke flow and smoke prevention and exhaust system; ⑧CFD analysis of boiler combustion (oil, gas, coal) rules; CFD analysis of exhaust hoods, CFD analysis of electrostatic precipitator, cyclone dust collector, and gravity-settling indoor air-particle separation process; ⑩ CFD of the interaction between urban wind (or micro-climate of the building district) and the quality of buildings and indoor air analysis. For the simulation of indoor air pollutant concentration distribution, foreign countries are in a leading position in this respect. Many large and excellent CFD softwares have been developed by foreign universities and research institutions. For example, in the United States, they used CFD to study the indoor distribution of indoor organic pollutants, that is, the numerical simulation of VOCs emissions. Research results in this area are also relatively rich. For example, the article Effect of ventilation pattern on room air and pollutant distribution published by Kee-Chiang Chung et al. In 2001 focused on the distribution of indoor pollutants.

In China, some software that can solve practical problems was also developed during the period, but in terms of simulation calculation of indoor humidity concentration distribution, applications are still lacking. At present, domestic indoor humidity is still mainly obtained by the previous actual point measurement method to obtain its distribution, but this method can only get some distribution results roughly, and can not obtain detailed distribution.

## 1.5. Description and prediction method of indoor temperature, humidity and pollutant concentration

The description and prediction methods of indoor non-uniform environmental parameters are usually divided into three types: theoretical calculation, model test and numerical simulation.

The traditional indoor air environment is usually predicted by jet theory calculations and model tests. The jet theoretical calculation has the advantages of quick and easy calculation and intuitive qualitative analysis, but most of them are only applicable to simple rooms and simple airflow patterns, and the scope of application is limited. When complex room shapes or more complex airflow organizations are encountered, the calculation is accurate Poor sex. Compared with theoretical calculations, model tests have the advantages of wide applicability and high reliability of results. With the development of computer imaging technology, in recent years, PIV, VPTV and other technologies have been applied to the analysis of indoor air flow experiments (Sun et al, 2007). In terms of model test research of indoor air parameters, some domestic and foreign scholars have also carried out corresponding research work, such as:

Cloth to calculate relevant evaluation indicators, such as air age and ventilation efficiency (Sandberg, 1981; Sandberg et al1983); and to study indoor pollutant transmission laws and describe distribution parameters of pollutants through model tests (Yang Jianrong, 2004; Cai Hao 2006; Chen Yan 2007). The model test can describe and analyze the real indoor environment. The data obtained is true and reliable, but the data obtained by the test method can only be values at some discrete points. There is a lack of understanding of the air parameter field distribution in the entire study area, so there is The result is insufficient information (Zhao Bin et al., 2001). Moreover, the test method is difficult to adapt to large spaces and complex systems when measuring dynamic processes. This is because the above systems have more disturbances and the dynamic processes are relatively long (Sandberg, 2007). In addition, there are disadvantages of long test cycle and high cost.

Since the late 20th century, with the development of computer technology, more and more scholars have begun to use computational fluid dynamics (CFD) numerical simulation methods to predict indoor air flow and distribution parameters such as temperature and pollutants , 1988; Nielsen, 1989; Jones et al, 1992; Braconnier et al, 2007), etc. Compared with theoretical calculations and model test methods, the numerical calculation method is low in cost, fast in cycle, good in visibility of calculation results, data Information-rich, indoor air loop

Forecasting of the environment has become a trend. In recent years, with the gradual deepening of related research, the simulation of CFD technology for indoor air environment has become more and more perfect (Zhai et al, 2007). However, when CFD numerical simulation encounters complex working conditions or a large number of calculations, especially when it is necessary to quickly obtain prediction results, there are still shortcomings of long calculation time. These shortcomings are rapid comparison of design solutions and optimization of indoor parameter control strategies It is particularly prominent when waiting for actual engineering problems.

In view of the shortcomings of the above prediction methods, some scholars have proposed many improved calculation methods. Such as region model method (Megri et al, 2007), dynamic region model method (Peng et al,1998), and FFD methods (Zuo et al, 2009). The area model can be used to quickly solve temperature, humidity, pollutant concentration, and other indicators that characterize the building characteristics, such as ventilation efficiency, energy consumption, pollutant transmission, and smoke diffusion. Be applicable. The FFD method is much faster than the traditional CFD method, but for complex spaces that require more computational grids, the advantages of the FFD method are not obvious. In addition, in order to improve the calculation efficiency and accuracy, some scholars have proposed a joint solution method for the distribution of multiple indoor air parameters (temperature, humidity, and pollutant concentration) based on CFD and boundary heat and mass transfer models (Wang et al, 2007; Samuel et al, 2006; Zhai et al, 2002). The above method solves the problems of multi-parameters and the coupling of heat transfer and flow, but it has the disadvantages of large amount of calculation and long calculation period. By adopting the various methods described above, the distribution of parameters such as indoor air speed, temperature, humidity, and pollutant concentration can be directly obtained, each of which has its advantages in form. However, its common disadvantage is that it cannot describe the degree of influence of various boundaries (such as air supply, source, etc.) on indoor environmental parameters, and it cannot reveal the distribution law of indoor environmental parameters. Therefore, when solving practical problems, it is usually only possible to use The traversal method is used to analyze a large number of working conditions.

In order to describe the distribution law of indoor air parameters in a non-uniform environment, some scholars have used the evaluation index method to conduct theoretical research on pollutant concentration, temperature, and humidity distribution characteristics under fixed flow field conditions. In terms of pollutant concentration, Murakami et al. Proposed a series of SVE indicators describing the diffusion characteristics and ventilation effectiveness of indoor pollutants under steady-state conditions (Murakami, 1992; Kato et al, 1994), and in addition, describing pollutant exclusion Efficient sewage or ventilation efficiency (Sandberg, 1983), dimensionless concentration describing the contribution of spatial concentration distribution (Tu Guangbei et al., 2002), spatial flow influencing factors describing the danger or immunity of flow field pollution (Zhang Yinping, 2006) And other indicators. In terms of temperature and humidity, Kato et al. Proposed a series of CRI indicators (Kato et al, 1994) on the relative magnitude of the intensity and range of the influence of heat sources under a defined airflow organization form (Kato et al, 1994). CRI indicators can be used to quantitatively measure the role of space in heat sources. Heat distribution and temperature change at a certain point in space. Huang et al. Proposed a CRI (H) indicator that describes the water vapor distribution law under the action of an internal moisture source under a fixed flow field condition based on the CRI concept (Huang et al, 2011). In addition, for the comprehensive evaluation index of indoor hot and humid environment, the International Standards Organization (ISO) published PMV and PPD indicators (ISO 7730, 1994). This indicator was proposed by Professor Fanger. It combines factors such as activities that affect human thermal sensation, clothing, air temperature, humidity, wind speed, and average radiant temperature. It has a high application value in evaluating the thermal comfort of indoor spaces (Olesen et al, 2002). Later, some scholars further improved this index, and proposed an IPMV index combined with the distribution density of people to evaluate the satisfaction of the hot and humid environment. Degrees (Meng Binbin, 2004).

The above indicators are not only used to evaluate the distribution of the temperature field, humidity field, and concentration field of the indoor air environment, but also used to determine the amount of fresh air and evaluate the air flow organization, which helps to obtain an overall evaluation of the performance of the ventilation and air conditioning system. However, the premise of the application of the above indicators is that the indoor concentration field, temperature field, and humidity field are all stable. In fact, compared with the stability of the air velocity field, the concentration, humidity, and temperature require longer propagation and transmission. Time can reach a steady state distribution in the air, so this requirement is far from the truth. Therefore, the above indexes are not suitable for the prediction and evaluation of the transient distribution of the above air parameters. In order to describe the dynamic process of transient indoor air parameters, some scholars have begun to study the propagation law of indoor pollutants. Li et al. proposed a series of accessibility indicators that reflect the propagation of passive gas pollutants under a fixed flow field, including supply air access and pollution source accessibility (Li et al, 2004).

The dynamic propagation process of pollutants under non-steady conditions has laid the foundation; Yang et al. Established a pollutant transmission relationship under the conditions that the initial concentration of indoor pollutants is evenly distributed and the walls are extinct, making the environmental estimation fast and easy. (Yang et al, 2004; Yang Jianrong, 2004). After that, Li et al. Considered the initial concentration distribution and wall boundary conditions that affect the dynamic propagation of pollutants, proposed the concepts of initial condition accessibility and boundary condition accessibility, and then gave the non-amorphous boundaries and constant boundary conditions. Relational expressions of pollutant propagation in a ventilated room at any initial condition (Li et al, 2008). The availability of a series of indicators laid the foundation for understanding the dynamic propagation of pollutants under unsteady conditions, and established a complete distribution and propagation relationship of the indoor mean concentration of pollutants under steady flow conditions and constant boundary conditions Formula, revealing the influence of various boundary conditions on the time-average of indoor pollutant concentration distribution, that is, the relationship can be used to quickly and easily solve the time-average of pollutant concentration at any indoor point, avoiding multiple iterations of traditional CFD The problem. In order to explore the contribution of various types of boundaries to the concentration of pollutants at any point in the room under non-constant boundary conditions, Li et al. Proposed a series of indicators of response coefficients under the condition of pulse release from pollution sources (Li et al, 2009). However, when used to solve the instantaneous value of pollutant concentration (such as online control, pollution source identification, etc.), the accessibility index is not suitable.

Therefore, if the solution is completely solved by the response coefficient index, when it encounters a large number of computational grids and complex working conditions with a long duration, it needs to store a large amount of data, so it has greater limitations. However, no one is currently engaged in the study of the instantaneous concentration distribution.

## 1.6. Design and control method of indoor non-uniform temperature, humidity and pollutant concentration environment

The ideal indoor air environment comes from good design and operation control. The traditional air conditioning system design and control method is a lumped parameter idea. Based on this idea, the indoor air is considered as a fully uniform mixture, and the indoor parameters are single parameters. Because of the spatial isolation, the parameters of each room do not interfere with each other, so the control focus is on the system level, and the corresponding research and engineering practice are more (Mathews et al, 2001; Nassif et al, 2005). In indoor non-uniform environments, the needs and control focus are on local environmental parameters (Jaakkola et al, 1989). Due to the non-uniformity of various indoor air parameters, many problems have arisen in actual operation control: (1) Due to the uneven distribution of environmental parameters, it is difficult to obtain real-time parameter values in the control area. A large number of engineering examples show that many problems are also caused by the out of control of local environmental control, among which the more prominent are the condensation on the inner surface of the envelope structure (Wang Hui et al., 2009; Ju Fali et al., 2011); (2) ) Because the spatial parameters are coupled with each other, the indoor control point oscillates; (3) Because multiple rooms are connected to each other, the return air outlet parameters are unknown, and it is more difficult to obtain the parameters of the control point, which often requires a large number of iterative calculations, which is far from the control requirements . In order to solve the control problem in the above non-uniform environment, some scholars have carried out corresponding research work. In the control of a non-uniform environment, it is necessary to meet the different parameter requirements of multiple locations at the same time, but because the air parameters between different locations have a large coupling relationship in the same space, the adjustment of a certain location parameter will inevitably affect The parameters of other surrounding locations, so when the requirements of different locations are simultaneously met, it is difficult to adjust in one step. In addition, more and more buildings are considering energy saving while ensuring air quality during operation. How to obtain the best operation control strategy according to actual needs is also an important issue. Therefore, it is difficult to control multiple locations in the same space. In the past, most of the automatic control of air parameters in air-conditioned and ventilated rooms used feedback adjustment methods, that is, when the air parameter (temperature, humidity, pollutant concentration) sensor passed the parameter to the calculator, and compared with the preset parameter value If it exceeds the control range, it will issue an instruction to the actuator of the air conditioning equipment to change the air supply parameters until the parameter requirements of the control point are met (Underwood, 1999). However, the biggest problem with feedback regulation is that it is prone to oscillations caused by overshoot (Franklin et al, 2002). Therefore, if the indoor control point parameters after the air supply parameters are changed can be accurately predicted, the occurrence of oscillation can be avoided. To solve the above problems, semi-feedforward and feedforward methods have emerged (Liu et al, 2002). However, semi-feedforward only presets several modes or calculates part of the workload, and in essence does not completely get rid of the feedback method; the current bottleneck of the feedforward method requires a lot of online calculations. In addition, some literatures use genetic algorithms to study the optimization control of VAV systems to solve the optimization of online strategies in multivariate situations (Wang et al, 2000), and some literatures use artificial neural network methods with feedforward properties ( Artificial neural networks (NNs) explored short-term predictions of air-conditioning, heating, and ventilation room loads (Kalogirou, 2000), and control of temperature and environmental parameters for precision air-conditioned rooms (An Dawei et al., 2004). It has significant contributions to reducing overshoot, anti-interference ability and improving temperature control accuracy, but the advantages of this method have not been systematically proven (Hippert et al, 2001). In addition, some scholars have adopted fuzzy control ideas to improve response speed and reduce steady-state errors (Li Chao et al., 2006). From the above, it can be seen that the feedforward adjustment method can maintain the adjustment accuracy and improve the control efficiency, which is the development trend of control methods for HVAC systems in the future, and also the development direction of indoor parameter control under non-uniform environmental conditions. Although in recent years, attempts to control the indoor air quality online using the CFD method have started to address the problem of controlling non-uniform environmental parameters (Ratnam et al, 1998), the large amount of calculations of the CFD simulation makes it only possible to use the simpler It can be applied in occasions. At present, few researches on control theory and corresponding control algorithms and strategies for non-uniform indoor environment have been carried out.

In addition to the control of the environmental parameters of a single room, actual ventilation and air-conditioning systems are often connected to multiple rooms, and due to the uneven distribution of air parameters (temperature, humidity, pollutant concentration, etc.) in the room, the exhaust parameters of the room cannot be obtained As a result, the air supply parameters of the system and the actual parameters of the room are unknown. When the system contains multiple air-conditioning boxes, the problem will be more complicated and the solution will be more difficult. Some scholars have used the method of the area-like model to calculate the age distribution of air in a room with return air, and their ideas are helpful for the calculation of concentration (Waters et al, 2000). For the situation where two air treatment equipments are connected to a room, some scholars have proposed the idea of using tracer gas method to determine the air supply volume, but have not paid much attention to the indoor concentration distribution (Roulet et al, 2003). Li et al. Proposed the concept of full air age and provided a direct solution algorithm to calculate the full air age of the room containing the return air (Li et al, 2003). However, in this solution process, it is assumed that the pollution sources in the room are evenly distributed in the whole space, which is quite different from the actual situation of pollution sources. In order to directly obtain the concentration of pollutants, some scholars proposed the concept of reference concentration, which largely solved the problem of directly solving the concentration of pollutants in the ventilation system with return air (Li Dongning, 2003; Li D N et al, 2004).

However, the reference concentration has not established a dimensionless relationship with the emission properties of the pollution source, so after the location or intensity of the pollution source changes, it must be calculated again using tools such as CFD, and the workload is still large. In 2004, Yang Jianrong and others reclassified the ventilation and air-conditioning system according to the different connection forms of the supply air and return air, so that the connection relationship can better establish a connection with the calculation of pollutant concentrations, and at the same time apply the availability of supply air and the source of pollution. The concept of accessibility establishes indoor concentration calculation relationships under different connection forms, which overcomes the shortcomings of traditional calculations that require multiple CFD iterations and lays the foundation for rapid calculations in engineering applications (Yang Jianrong, 2004). Based on this, Zhu Fenfei et al. Used the concept of response coefficients from the pulse emission of pollution sources, and gave the calculation formulas of indoor pollutant concentration under different transmission system connection forms (Zhu Fenfei, 2009). The indoor concentration calculation problem under varying boundary conditions, but this method is all based on the response coefficients. When a system with a large number of complex rooms is required, massive data storage is required, and its applicability is limited. At present, the dynamic calculation methods for the distributed parameters in various rooms in complex air-conditioning systems need to be improved and perfected.

## 1.7. Literature review summary

Creating an indoor non-uniform air environment is an important means to improve indoor air quality and reduce building energy consumption. It is of great significance to improve the quality of living environment and the efficiency of ventilation and air conditioning systems. Therefore, research and engineering practice around indoor non-uniform environmental parameters have become a hot spot at home and abroad. Temperature, humidity, and the concentration of gaseous pollutants are the three most important indicators to characterize indoor environmental parameters. Therefore, research on indoor environmental parameters has mostly been carried out around these three indicators. The related research results on the temperature, humidity, and pollutant concentration parameters in a non-uniform indoor environment are summarized as follows:

(1) The description and prediction of the temperature, humidity, and concentration of gaseous pollutants in an indoor heterogeneous environment are the primary content of the creation of a heterogeneous environment. The prediction methods are usually divided into three types: theoretical calculation, model test and numerical simulation. Theoretical calculation has the advantages of quick and easy calculation and intuitive qualitative analysis, but most of them are only applicable to simple rooms and simple airflow patterns, and the scope of application is limited; the data obtained from model tests are true and reliable, but there is a lack of insufficient information and long test cycles The cost is high; the method of numerical calculation has low cost, fast cycle, good visibility of the calculation result, and rich data information, but when it encounters complex working conditions or a large number of calculations, especially when it is necessary to quickly obtain the prediction result, it is still There is a disadvantage of long calculation time. The shortcomings of the above methods are, in the final analysis, the lack of understanding of the distribution law of non-uniform indoor parameters. In order to describe the distribution law of indoor air parameters in non-uniform environments, some scholars have used the evaluation index method to conduct theoretical research on pollutant concentration, temperature, and humidity distribution characteristics under fixed flow field conditions, but these methods have not taken into account the dynamic process. Description of the transient parameters in. In engineering design and operation control related to non-uniform environment, it is more necessary to grasp the instantaneous parameters, so it is urgent to conduct research on the instantaneous distribution of air parameters (including temperature, humidity, and concentration of gas pollutants). .

(2) In the field of HVAC, the traditional indoor environmental parameter control is based on the idea of lumped parameters, so it cannot meet the control needs of non-uniform environmental parameters. In the control of non-uniform environment, it is usually necessary to meet the different parameter requirements of multiple locations at the same time, but because the air parameters between different locations in the same space have a large coupling relationship, when the feedback adjustment strategy is adopted, it is bound to The oscillation caused by overshoot is difficult to adjust in one step during the adjustment process. At present, there is no good solution found in related research and engineering practice. In addition, more and more buildings are considering energy saving while ensuring air quality during operation. How to obtain the best operation control strategy according to actual needs is also a new hotspot in recent years. Therefore, to guarantee multiple control needs in the same space, it is urgent to make

breakthroughs in control algorithms and strategies.

(3) In actual ventilation and air-conditioning systems, multiple rooms are often connected, and due to the uneven distribution of air parameters (temperature, humidity, pollutant concentration, etc.) in the room, the exhaust parameters of the room cannot be obtained, which causes the system to be delivered. The wind parameters and the actual parameters of the room are unknown. When the system contains multiple air-conditioning boxes, the problem will be more complicated and the solution will be more difficult. Some scholars have proposed a variety of solutions to solve the indoor concentration calculation problem under arbitrary changing boundary conditions in the return air system, but the practical applications of these methods have limitations. At present, the dynamic calculation methods of the distributed parameters in various rooms in complex air-conditioning systems still need to be broken and improved in terms of calculation theoretical models.

## 1.8. Composition

The paper composition showed on figure 1-3.

This paper, firstly, expands the CFD calculation principle and method from the perspective of mathematical theory, and then determines the feasibility of this technology for indoor air temperature and humidity simulation based on this theory. Based on this method, in the thereinafter chapter, this paper carried out CFD simulation of laboratory temperature and humidity in combination with actual laboratory conditions. The content of this paper are as follows: In order to study the effect of different ventilation methods on the indoor humidity distribution and the formation of indoor condensation conditions, a simple physical model was established under the existing mathematical model, and CFD simulation was performed on the humidity distribution in each model under different ventilation methods using CFD technology. The results show that the air supply and return air outlets are on both sides of the room and the bottom-in and back-up method is the best way to control the indoor air humidity and greatly reduce the occurrence of dew condensation.

According to the simulation results, in the case of the room ventilation method, a small single apartment is taken as the research object, and the effect of the humidifier placement and height on the indoor humidity distribution in winter was studied. And get the result that at a height of 0.5 meters, condensation on the ground rarely occurs.

At present in Japan, due to the internal activities of human and the heat dissipation of various equipment in office buildings, more and more office buildings use air-conditioning for cooling in winter, and due to the cooling system, the supply air cannot be humidified during cooling. Indoor humidity conditions are not comfortable. This article simulates the indoor temperature and humidity of the office building refrigeration system in winter. It is hoped that the research results can guide the control decision of actual indoor temperature and humidity.

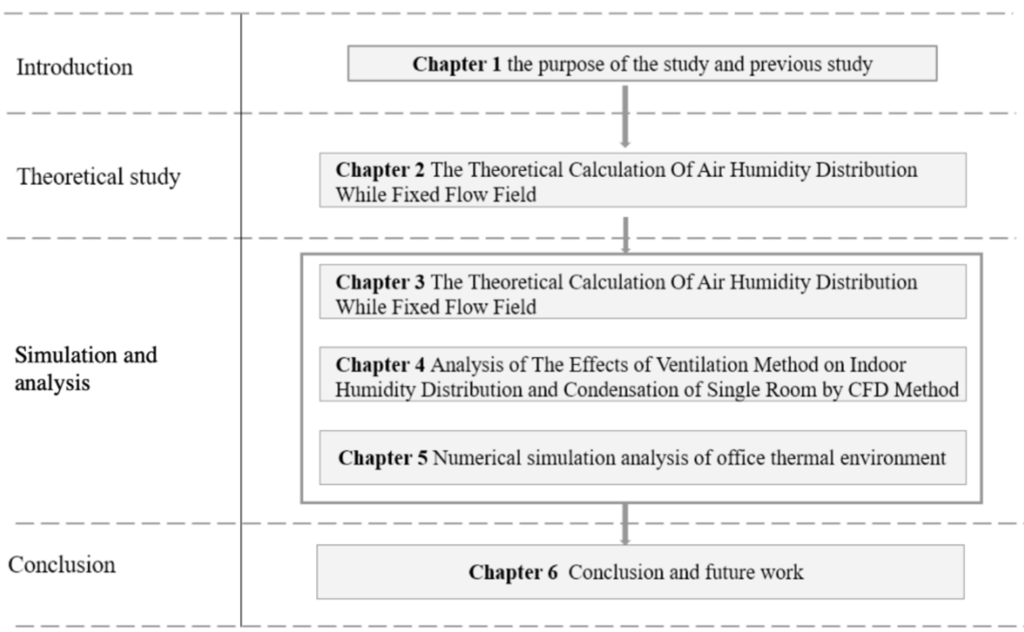



Figure 1-3 Paper Composition

# CHAPTER 2

# THE THEORETICAL CALCULATION OF AIR HUMIDITY DISTRIBUTION WHILE FIXED FLOW FIELD

# 2. The theoretical calculation of air humidity distribution while fixed flow field

## 2.1. Introduction

Indoor environment is an important place that closely related people's daily work and life. Humidity is an important indicator of indoor air parameters. And Japan is accelerating into an aging society. More and more elderly start using welfare facilities. In winter time, heater is used frequently. The influence of ventilation and humidify can not be ignored. If the winter indoor air is too dry, it will be a great influence to the health of living, especially for the elderly. In order to improve the quality of life in the elderly, the improvement of the living environment is indispensable. Most of the elderly can know the indoor temperature regime through thermometer, but they always ignore the indoor humidity regime. The elderly is insufficient understanding of the importance of the indoor humidity. In addition, Indoor humidity conditions are strongly associated with the survival rate of virus. So the humidity control is particularly important.

In order to describe the contribution of air, humidity and initial distribution to the indoor moisture content distribution. The concept of transient accessibility is introduced. Including the transient accessibility of supply air, the transient accessibility of wet source and the transient accessibility of initial condition. So that, on this basis, then obtained the algebraic equation between the instantaneous moisture content and the influence factors at any time.

## 2.2. Ventilation room water vapor transmission principle

In the building indoor environment, generally the water vapor in the humid air is in a superheated state, and the content is very low, so at a certain extent it can be approximated as the ideal gas. At the same time the density is small, the diffusion of wet air can be ignored. Therefore, about the physical process the propagation of wet air can be treated as passive gas, and it observe the component propagation control equation.

In passive gas transport equation, the equivalent mass diffusion coefficient is:

$$\Gamma_{Ceff} = \frac{\mu}{\sigma} + \frac{\mu_t}{\sigma_t} \tag{2-1}$$

For the indoor air flow range, the first term in the right-hand side of the formula, Laminar viscosity coefficient$\mu$ Usually the value is $1.8 \times 10^{-5}$ $(\frac{\mathrm{kg}}{\mathrm{ms}})$ . But the mass transfer Schmidt number is related to the species.

$$\sigma = \frac{\upsilon}{\mathrm{D_{AB}}} \tag{2-2}$$

The kinematic viscosity of air is usually taken $2.5 \times 10^{-5}$ $(m^2/s)$ , Diffusion coefficient $D_{AB} = 2.5 \times 10^{-5}(m^2/s)$. The second term in the right-hand side of the formula, The order of magnitude of turbulent viscosity coefficient$\mu_t$always taken $10^{-1}$~$10^{-2}$.. Turbulent mass transfer Schmidt number $\sigma_t$ always be 1. Combine (2-1) and (2-2), the first term on the right-hand side of the formula is far less than the second term and therefore can be ignored in the calculation.

So：

$$\Gamma_{Ceff} = \frac{\mu_t}{\sigma_t} \tag{2-3}$$

In summary， water vapor transport equation：

$$\frac{\partial_\rho C(\tau)}{\partial_\tau} + \frac{\partial_\rho C(\tau) U_j}{\partial_{x_j}} = \frac{\partial}{\partial_{x_j}}(\Gamma_{ceff}\frac{\partial C(\tau)}{\partial_{x_j}}) + S(\tau) \tag{2-4}$$

The boundary conditions and initial conditions of the corresponding humidity：

The boundary conditions: $\begin{cases} C(\tau) = C_S^{n_s} & On\ the\ n_s\ th\ airflow\ outlet \\ S(\tau) = J^{n_c} & On\ the\ n_c th\ wet\ source \end{cases}$ (2-5)

Initial conditions: $C(\tau)|\tau = 0 = C^P(0)$ (2-6)

When the airflow in the room is stabilized, the change in moisture content in the air at any point in the room is affected by the initial distribution of the airflow, internal moisture, and the moisture content in room.

As shown in Figure 2-1, when the steady state is reached, the concentration of the point P in the room is affected by all the effects (S1 and S2) and the source (internal source C1 and side wall source C2); and in the transient case. In addition to the contribution of air supply and source, there is an impact from the initial concentration distribution I of indoor environment.

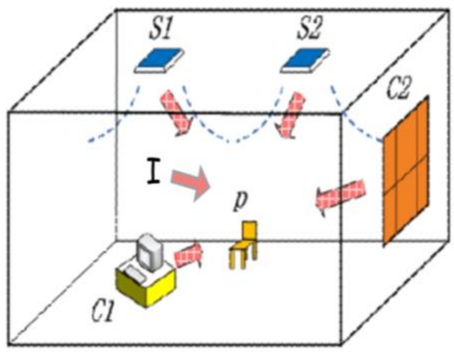



Figure 2-1 The influencing factors of indoor water vapor concentration

When the flow field is determined and remains stable, equation (2-4) is a linear equation, due to the superposition principle, the moisture content at any point P in the chamber can be obtained from the sum of the linear algebra of the air supply, the wet source and the initial distribution. Under these conditions, the boundary conditions and initial conditions of Equations (2-5) and (2-6) can be decomposed into three parts:

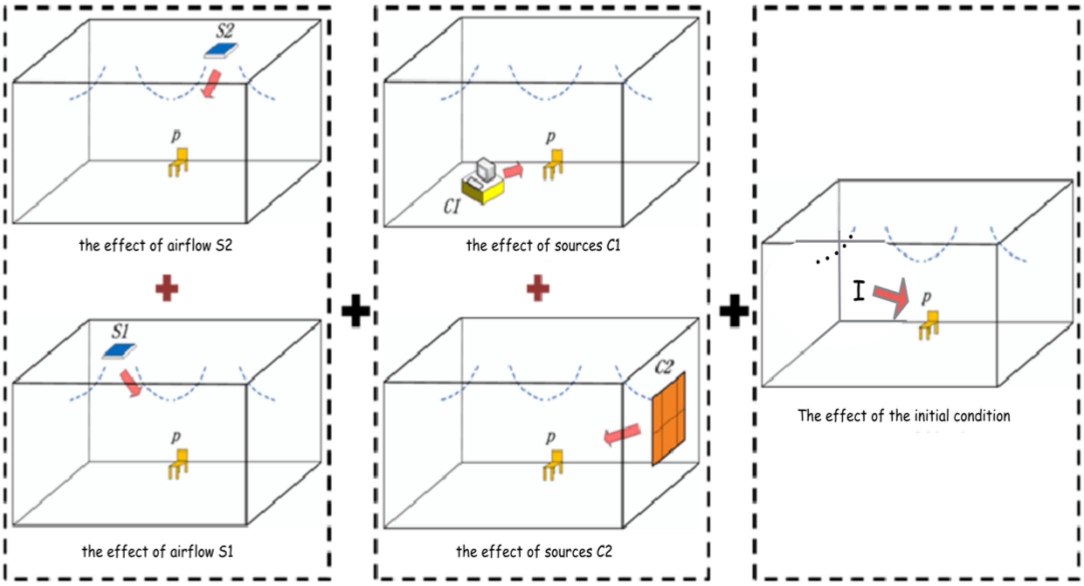



Figure 2-2 Indoor instantaneous concentration superposition principle schematic

1. The effect of air supply:

The boundary conditions: $\begin{cases} C(\tau) = C_S^{n_s} & On\ the\ n_s\ th\ airflow\ outlet \\ S(\tau) = 0 & On\ all\ wet\ source \end{cases}$ (2-7)

Initial conditions: $C(\tau)|\tau = 0 = 0$ (2-8)

2. The effect of sources

The boundary conditions : $\begin{cases} C(\tau) = 0 & On\ all\ airflow\ outlet \\ S(\tau) = J^{n_c} & On\ the\ n_c th\ wet\ source \end{cases}$ (2-9)

Initial conditions: $C(\tau)|\tau = 0 = 0$ (2-10)

3. The effect of the initial condition

The boundary conditions: $\begin{cases} C(\tau) = 0 & On\ all\ airflow\ outlet \\ S(\tau) = 0 & On\ all\ wet\ source \end{cases}$ (2-11)

Initial conditions: $C(\tau)|\tau = 0 = C^P(0)$ (2-12)

When the room flow field is fixed, The moisture content at any point in the room is expressed as the contribution of each boundary and the initial conditions and other factors at this moment. In order to describe the contribution of each influencing factor, the degree of influence of each boundary and initial condition at a certain moment is defined as the transient accessibility of the moment.

In order to describe the contribution of air, moisture, and initial distribution to the indoor moisture content distribution, respectively. The concept of transient accessibility is introduced. Including the transient accessibility of supply air, the transient accessibility of wet source and the transient accessibility of initial condition. So that, on this basis, then obtained the algebraic equation between the instantaneous moisture content and the influence factors at any time

## 2.3. The concept of transient accessibility.

For a fixed flow field, there is no indoor emission source and the wall is adiabatic, the initial concentration is zero. When a certain outlet ($n_s$) from the 0th time constant release of a certain concentration (C),The air transient accessibility at any arbitrary point (P) ,at any time ($\tau$) in the space is defined as follows:

$$a_s^{n_{s,p}}(\tau) = \frac{d^P(\tau)}{d^{n_s}} \tag{2-13}$$

Where: $d^{n_s}$ is the $n_s$th air outlet air moisture content;

Transient accessibility of supply air (TASA) is a dimensionless number. Reflecting the various air outlets on the room instantaneous moisture content of any point of the degree of impact. If a tuyere of a point on the indoor TASA larger, then the outlet on the instantaneous moisture content of the greater the extent. TASA is a time-related indicator, the value of the room only by the air organization, and has nothing to do with the air supply concentration. The transient accessibility of supply air reflects the inherent properties of the flow field. It is value can be obtained by actual measurement or simulation of indoor air moisture content.

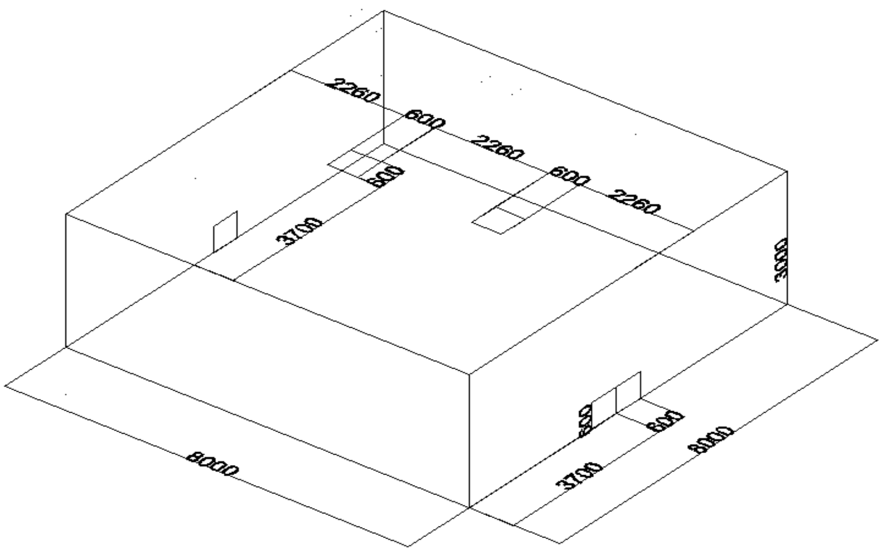



Figure 2-3 Room layout

In the following, the transient accessibility of supply air will be described with a specific example. Figure is an air supply room, room size is X (length) ×Y (H) ×Z (W) =.8×3×8。 On the top of the room there are two air outlets (0.6m×0.6m) labeled S1, S2; on the side wall there are two return air outlets (0.6m×0.6m) marked E1, E2. The position and tuyere size are detailed in the figure. Room ventilation every 10 minutes, each outlet wind speed of 5 meters per second. There is no heat source in the room， and the walls are insulated. Indoor flow field stability, cross-section velocity vector distribution shown in Figure. The calculation parameters are as follows: S1, S2 export speed is 5m / s, temperature is 298K. The water vapor concentration is 60%.

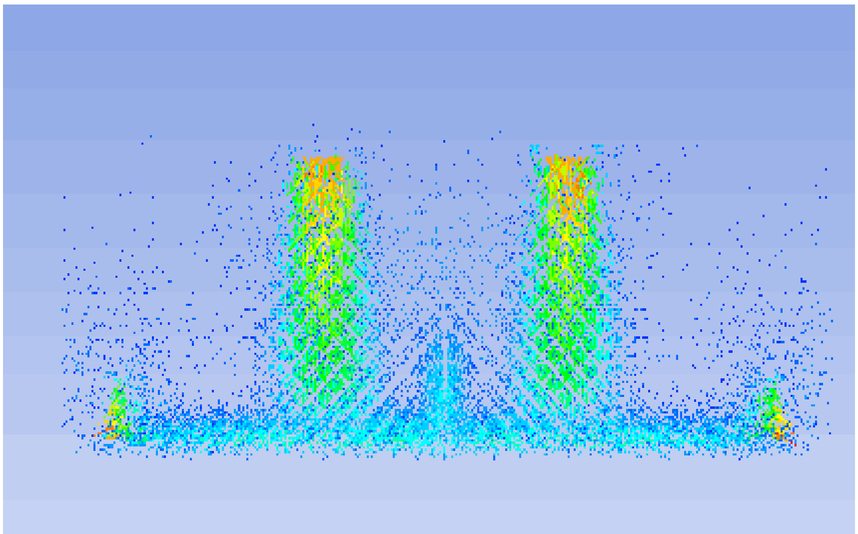
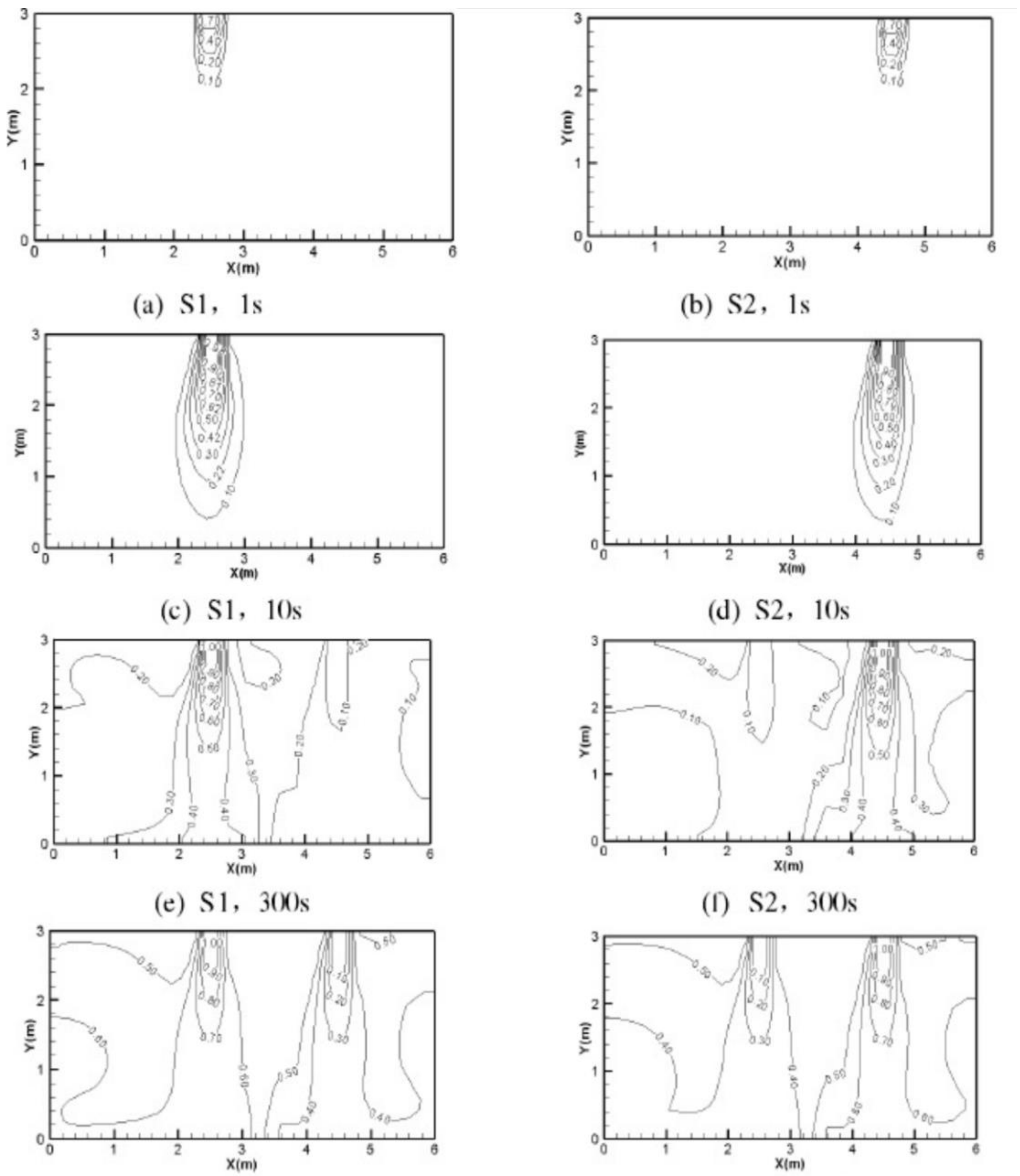


Figure 2-4 Velocity vector distribution(z=1.5)

Figure shows the instantaneous accessibility of the tuyere S1 and S2 at each moment. The degree of air supply reflects the degree of influence of each air supply on the different positions of the room in the case of the flow field. Take the air outlet S1 as an example, it can be seen from the figure when the indoor flow field is stable, starting from the 0th hour, the beginning of the air supply, the first second in addition to the outlet near the relatively large value of the larger, most of the space Of the degree of accessibility are 0, that is, the air supply to most parts of the room has no effect (Figure 2-5a). As time goes on, the range of accessibility is gradually widened, indicating that the influence of air supply on the room is expanding (Figure 2-5c, e). When the arrival of 3000 seconds, the indoor instantaneous degree of distribution has been no change, indicating that the humidity distribution to steady state, this time from the figure (2-5g) can be clearly seen in the air outlet S1 near the outlet. The value of 1, indicating that the air moisture content here is equal to S1 air in the air moisture content. In the process of arriving at steady state from time 0, the influence of the air supply port S1 on the room can be gradually seen from the distributable distribution. However, the degree of accessibility near S2 is the smallest in space. The nearby accessibility value is equal to zero, so it can be concluded that the S1 air supply has the least effect on the area near the S2 outlet in the room, or even 0, which is almost entirely affected by S2. Figure 1-5 shows the similarity of S2 and S1 for the distribution of the air supply in the air supply vane S2 over time. When the steady state is reached, the instantaneous availability of all the air supply (S1 and S2) is added, and the degree of accessibility of all the spaces is 1, and it can be concluded that when there is only air supply in the room, Air moisture content comes from the superposition of the effects of each air supply. The availability of air can be easily analyzed the impact of the various air supply on the indoor range and the extent of the impact of the entire air supply conditions under the indoor moisture content of environmental characteristics.




(a) S1， 1s (b) S2， 1s

(c) S1， 10s (d) S2， 10s

(e) S1， 300s (f) S2， 300s

Figure 2-5 Different air outlets at all times the transient accessibility of supply air distribution

It can be seen from the above description, using the model to obtain the transient availability of air outlet and indoor humidity distribution, can be more easily analysis the various air supply on the indoor humidity impact and the degree of influence, as well as the indoor air humidity environment characteristics of the air supply conditions.

## 2.4. Transient accessibility of wet source (TAWS)

For a fixed flow field, there is no air conditioning and the wall is adiabatic, the initial concentration of each inlet concentration and rooms were 0. When a certain wet source ($n_c$) from the 0th time constant release of a certain distribution intensity $J_d^{n_c}$ ,The wet source transient accessibility at any arbitrary point (P) ,at any time ($\tau$) in the space is defined as follows:

$$a_c^{n_{c,p}}(\tau) = \frac{d^P(\tau)}{d_E^{n_c}} ,d_E^{n_c} = \frac{J_d^{n_c}}{Q} \tag{2-14}$$

Where: $J_d^{n_c}$ is the distribution intensity of $n_c$th wet source ;( g/s)

$d_E^{n_c}$ is the average concentration of the inlet at steady state ; (g/kg)

Q is the room ventilation mass flow. ($\rho \times kg$/s)

Transient accessibility of wet source (TAWS) also be a dimensionless number. Reflecting the degree of influence of each wet source on the instantaneous concentration of any point in the room. Source accessibility is a time-related indicator.

When the source location within the room is determined, its value is determined only by the airflow within the room, regardless of the concentration of the source itself. It is value can be obtained by actual measurement or simulation of indoor air moisture content.

Figure 2-6 is an air supply room, room size is X (length) ×Y (H) ×Z (W) =.8×3×8。 On the top of the room there are two air outlets (0.6m×0.6m) labeled S1, S2; on the side wall there are two return air outlets (0.6m×0.6m) marked E1, E2. In the example room, add two sources C1 (2560,3700,1000), C2 (5420,3700,1000), as shown. The distribution intensity of C1 and C2 is different. The calculation parameters are as follows: S1, S2 export speed is 5m / s, temperature is 298K. The outlet gas is dry air. And the wet source C1 export speed of 5m / s, distribution intensity of 100g / s.; C2 outlet speed of 5m / s, the distribution intensity of 200g / s.

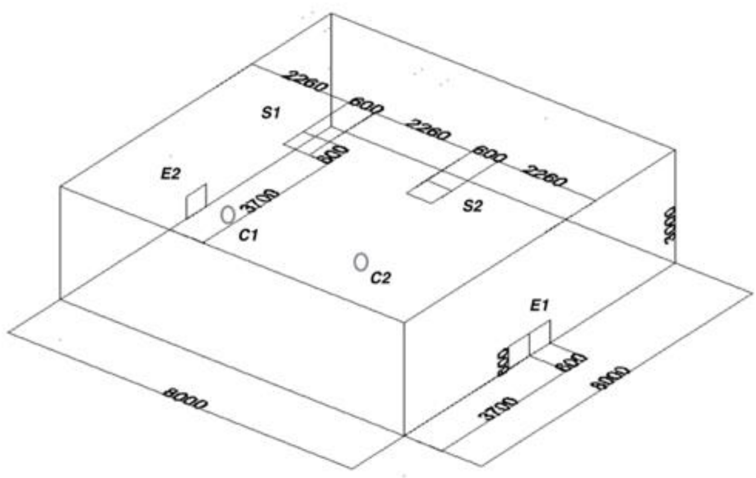



Figure 2-6 Room layout

Assuming that the volume of the source can be ignore, the water vapor as passive gas. Assuming that the source has no effect on the flow field during the distribution process.

The position and tuyere size are detailed in the figure. Room ventilation every 10 minutes, each outlet wind speed of 5 meters per second. There is no heat source in the room， and the walls are insulated. Indoor flow field stability, cross-section velocity vector distribution shown in Figure 2-7.

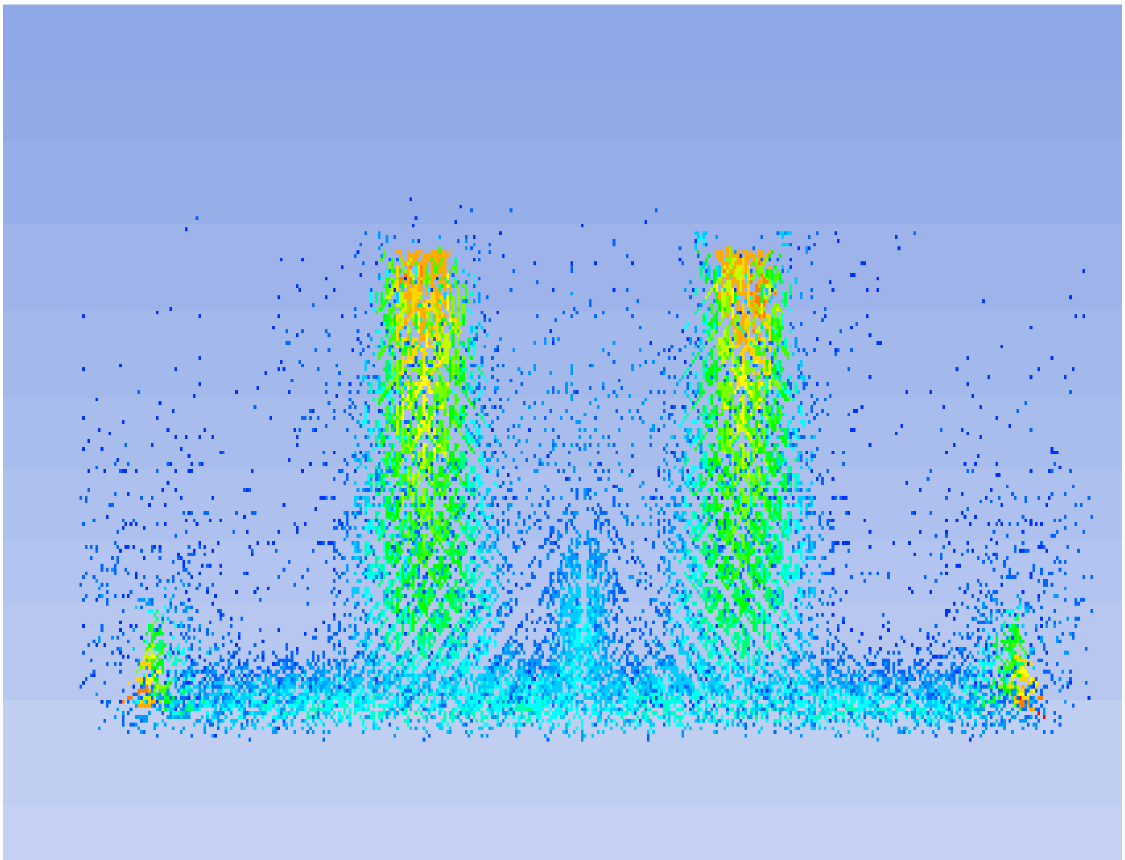


Figure 2-7 Velocity vector distribution(z=1.5)

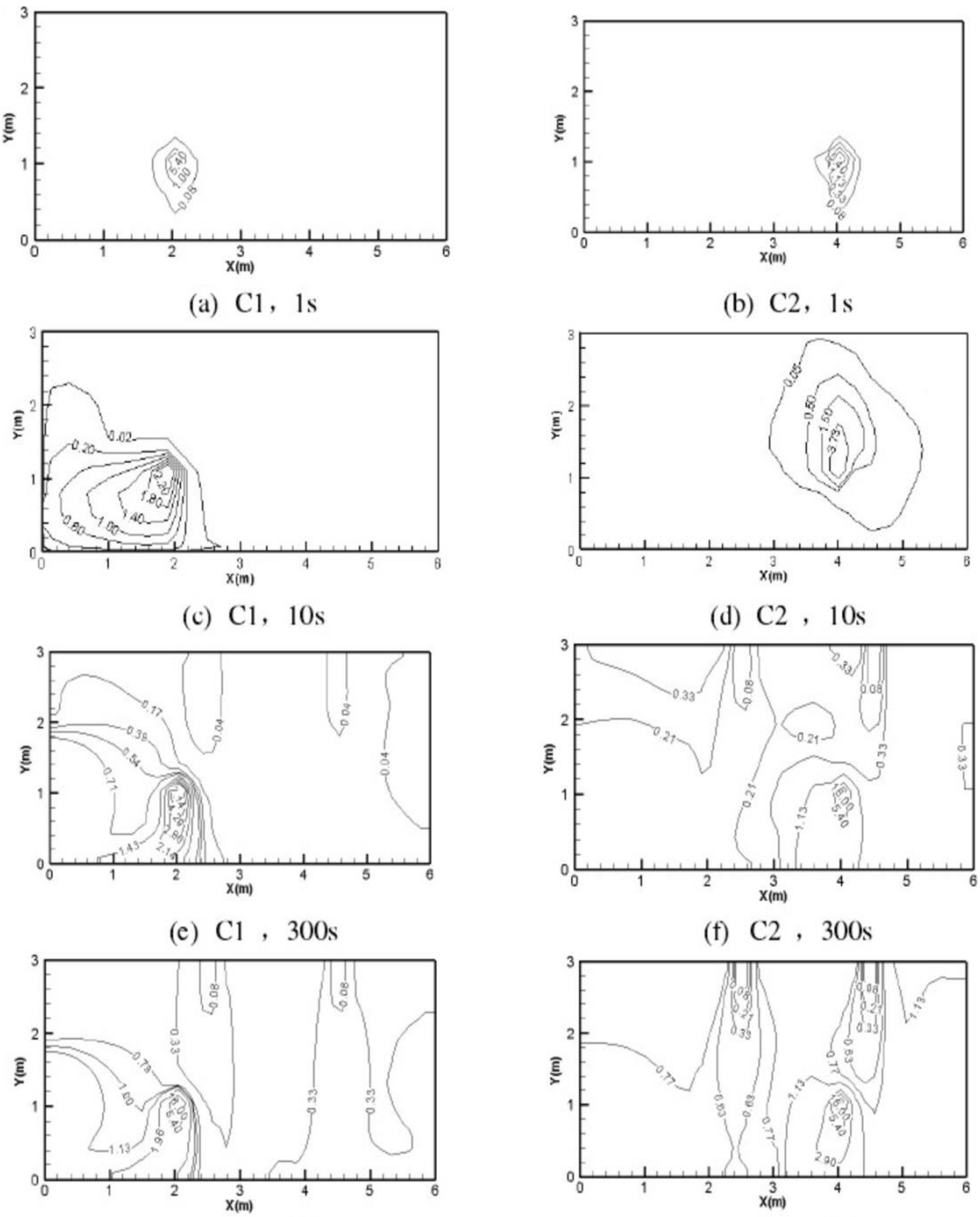



Figure 2-8 Different wet source at all times the transient accessibility of wet source distribution

The figure 2-8 shows the instantaneous accessibility of the source C1 and C2 at each moment. It can be seen that the distribution of accessibility is closely related to the position of the source at the beginning of the distribution, and the effect of the flow field is not obvious (Figure2-8. A, b, c, d). When the time reaches 300s, it is clear from the figure that the effect of the flow field begins to appear (Figure2-8 e, f). When the steady state (3000s) is reached, can be seen, the inlet on the distribution of the degree of impact also had a great impact. In summary, the combined effect of the spread of water vapor in the room is by the position of the source and flow field.

## 2.5. Transient accessibility of initial condition (TAIC)

For a fixed flow field, there is no wet source and the wall is adiabatic, each airflow inlet concentration was 0. At the 0th time there is an initial distribution of wet air in the room, then at any time thereafter (τ), the transient accessibility of initial condition (TAIC), at any arbitrary point(p) is defined as follows:

$$a_I^p(\tau) = \frac{d^P(\tau)}{\overline{d_0}}, \overline{d_0} = \frac{\oint d^P(0)dV}{V} \tag{2-15}$$

Where : $\overline{d_0}$ is the average of the moisture content of the air in the room at the 0th time.

Transient accessibility of initial condition (TAIC) also be a dimensionless number. Reflecting the conditions in a certain flow field, the degree of influence on the initial humidity distribution of the room at any time during a continuous period of time. The initial condition can be determined by the initial distribution of humidity and the organization of airflow within the room,

Regardless of the specific value of the initial concentration. Therefore, the initial condition can reflect the distribution of the initial conditions of the room itself. When the source location within the room is determined, its value is determined only by the airflow within the room, regardless of the concentration of the source itself. It is value can be obtained by actual measurement or simulation of indoor air moisture content.

In this part, the calculation parameters are as follows: the temperature is 298K and the initial humidity is 40%. And it can be seen that at the beginning, the initial distribution works on the space that it occupied. Due to the influence of the air supply, the transient accessibility of initial condition of the position near the air inlet is lower. When the steady state is reached, the effect of the initial distribution on the interior is almost non-existent, and transient accessibility of initial condition is closet zero in the whole space.

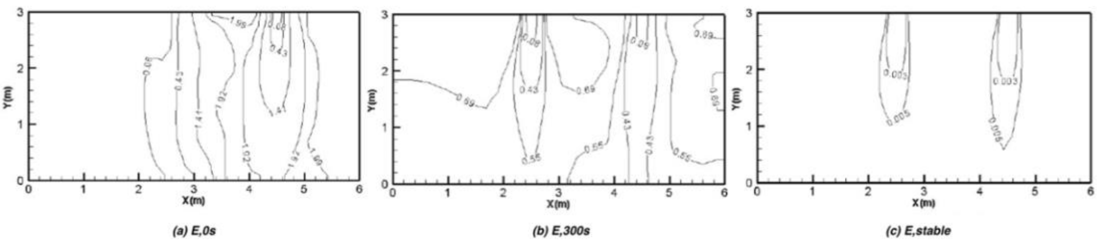



Figure 2-9 Initial condition at all times the transient accessibility of initial condition distribution

## 2.6. The theoretical calculation of air humidity distribution while fixed flow field

Therefore, according to the principle of linear superposition, when the flow field conditions determined, can be obtained in a variety of fixed boundary conditions at the same time. The instantaneous moisture content d(g/kg) of any point (p) at time(τ) can be expressed as:

$$d^P(\tau) = \sum_{n_s=1}^{N_s}\left[d^{n_s} a_s^{n_s,p}(\tau)\right] + \sum_{n_c=1}^{N_c}\left[\frac{J_d^{n_c}}{Q}\ a_c^{n_c,p}(\tau)\right] + \overline{d_0}\ a_I^p(\tau) \qquad (2\text{-}16)$$

When the state is stable, the contribution of the initial condition is 0, and the indoor humidity distribution is affected by the source, source and boundary conditions:

$$d^P(\infty) = \sum_{n_s=1}^{N_s}\left[d^{n_s} a_s^{n_s,p}(\infty)\right] + \sum_{n_c=1}^{N_c}\left[\frac{J_d^{n_c}}{Q}\ a_c^{n_c,p}(\infty)\right] \qquad (2\text{-}17)$$

## 2.7. Transform absolute humidity to relative humidity:

First, In the formula “d” is the absolute humidity. The absolute humidity between the mass of water vapor present in moist air to the mass of dry air. Absolute humidity is normally expressed in kilograms of water vapor per kilogram of dry air.

Absolute humidity expressed by mass:

$$\mathrm{d} = \frac{\mathrm{m_w}}{\mathrm{m_a}} \qquad (2\text{-}18)$$

Where: d = absolute humidity;

$m_w$= mass of water vapor (kg):

$m_a$= mass of dry air (kg)

## 2.8. Wet air enthalpy chart：

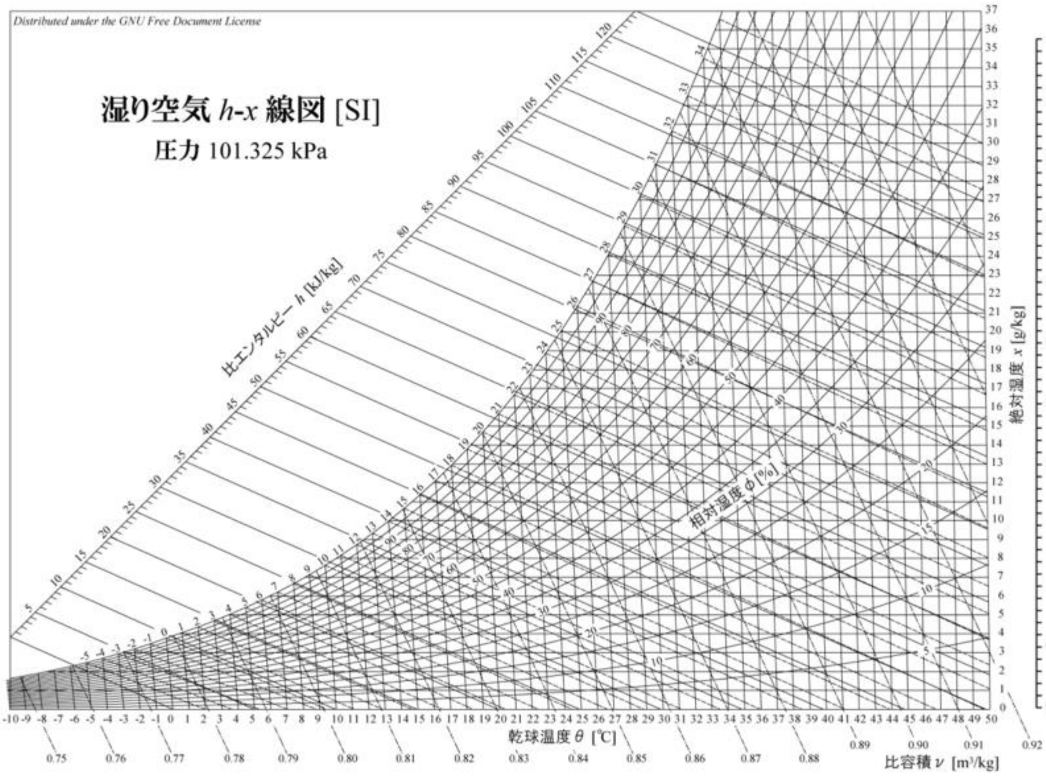



Figure 2-10 Wet air enthalpy diagram

Wet air enthalpy chart is referred to as a pneumatic diagram or humidity diagram. In the chart include dry bulb temperature, wet bulb temperature, absolute humidity, relative humidity, dew point temperature and enthalpy. And it can find all of parameters easily by setting any two values.

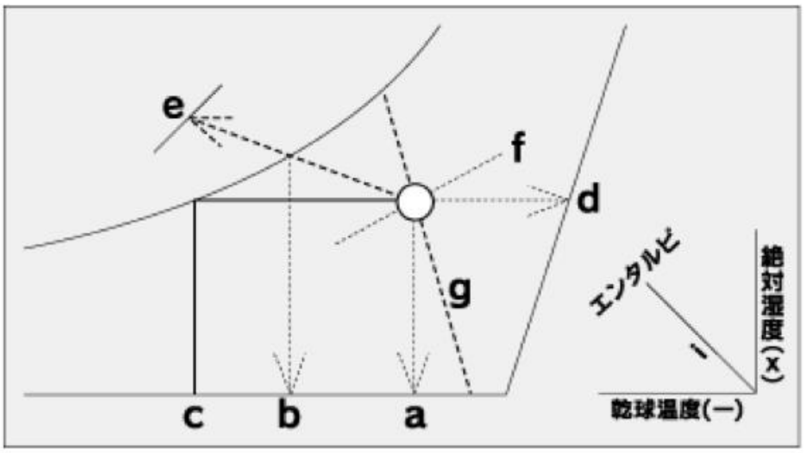



Figure 2-11 The explanation of wet air enthalpy diagram

And the wet air enthalpy diagram shows the relationship between the state of humid air, temperature and enthalpy. In the figure, the horizontal axis is temperature (a,b,c),the enthalpy on the slant side axis (e),and the absolute humidity on the vertical axis (d), and ‘f’ is relative humidity. When two of the state values of the humid air are determined, all other status values are found on the diagram.

## 2.9. Though pressure saturated air moisture content.

$$\varphi = \frac{d}{db} \times 100\% \qquad (2\text{-}19)$$

Table 2-1 0.101325MPa (a) pressure saturated air moisture content(db):

| Air temperature ℃ | Saturated water vapor partial pressure MPa | Moisture content g/kg |
|---|---|---|
| -40 | 0.0000128 | 0.080 |
| -35 | 0.0000223 | 0.139 |
| -30 | 0.0000380 | 0.236 |
| -29 | 0.0000421 | 0.262 |
| -28 | 0.0000467 | 0.291 |
| -27 | 0.0000517 | 0.322 |
| -26 | 0.0000572 | 0.356 |
| -25 | 0.0000632 | 0.394 |
| -24 | 0.0000699 | 0.435 |
| -23 | 0.0000771 | 0.480 |
| -22 | 0.0000850 | 0.529 |
| -21 | 0.0000937 | 0.583 |
| -20 | 0.0001032 | 0.643 |
| -19 | 0.0001135 | 0.707 |
| -18 | 0.0001248 | 0.777 |
| -17 | 0.0001371 | 0.854 |
| -16 | 0.0001506 | 0.938 |
| -15 | 0.0001652 | 1.029 |
| -14 | 0.0001811 | 1.128 |
| -13 | 0.0001984 | 1.237 |
| -12 | 0.0002172 | 1.354 |
| -11 | 0.0002376 | 1.481 |
| -10 | 0.0002597 | 1.620 |
| -9 | 0.0002837 | 1.770 |
| -8 | 0.0003007 | 1.876 |
| -7 | 0.0003379 | 2.109 |
| -6 | 0.0003625 | 2.263 |
| -5 | 0.0004015 | 2.507 |
| -4 | 0.0004372 | 2.731 |
| -3 | 0.0004757 | 2.973 |
| -2 | 0.0005173 | 3.234 |
| -1 | 0.0005623 | 3.517 |
| 0 | 0.0006108 | 3.823 |

| Air temperature ℃ | Saturated water vapor partial pressure MPa | Moisture content g/kg |
|---|---|---|
| 12 | 0.001403 | 8.851 |
| 13 | 0.001498 | 9.459 |
| 14 | 0.001599 | 10.107 |
| 15 | 0.001706 | 10.795 |
| 16 | 0.001819 | 11.524 |
| 17 | 0.001938 | 12.293 |
| 18 | 0.002064 | 13.109 |
| 19 | 0.002198 | 13.979 |
| 20 | 0.002339 | 14.897 |
| 21 | 0.002488 | 15.870 |
| 22 | 0.002645 | 16.899 |
| 23 | 0.002810 | 17.984 |
| 24 | 0.002985 | 19.138 |
| 25 | 0.003169 | 20.356 |
| 26 | 0.003363 | 21.646 |
| 27 | 0.003567 | 23.007 |
| 28 | 0.003782 | 24.449 |
| 29 | 0.004008 | 25.971 |
| 30 | 0.004246 | 27.581 |
| 31 | 0.004495 | 29.275 |
| 32 | 0.004758 | 31.073 |
| 33 | 0.005034 | 32.971 |
| 34 | 0.005323 | 34.971 |
| 35 | 0.005627 | 37.087 |
| 36 | 0.005945 | 39.315 |
| 37 | 0.006280 | 41.679 |
| 38 | 0.006630 | 44.167 |
| 39 | 0.006997 | 46.796 |
| 40 | 0.007381 | 49.568 |
| 41 | 0.007784 | 52.503 |
| 42 | 0.008205 | 55.597 |
| 43 | 0.008846 | 60.362 |

| | | | | | |
|---|---|---|---|---|---|
| 1 | 0.000657 | 4.115 | 44 | 0.009108 | 62.329 |
| 2 | 0.000700 | 4.385 | 45 | 0.009590 | 65.977 |
| 3 | 0.000758 | 4.751 | 46 | 0.010090 | 69.803 |
| 4 | 0.000814 | 5.102 | 48 | 0.011170 | 78.214 |
| 5 | 0.000873 | 5.475 | 50 | 0.012340 | 87.560 |
| 6 | 0.000935 | 5.873 | 51 | 0.012970 | 92.696 |
| 7 | 0.001002 | 6.296 | 52 | 0.013620 | 98.074 |
| 8 | 0.001073 | 6.746 | 53 | 0.014300 | 103.788 |
| 9 | 0.001148 | 7.223 | 54 | 0.015010 | 109.851 |
| 10 | 0.001228 | 7.733 | 55 | 0.015750 | 116.279 |
| 11 | 0.001313 | 8.276 | 56 | 0.016520 | 123.089 |
| 12 | 0.001403 | 8.851 | 57 | 0.017320 | 130.298 |

## 2.10. Simulation feasibility analysis

The indoor simulation experiment was conducted from November 16th to December 11th, 2015 at a laboratory N312-3 (6m*8m*2.5m) in Kitakyushu University by Lu Xin.
In this laboratory, room is divided into two spaces, one is main laboratory, the other is seminar room that connects outside directly. There are 14 students, 15 computers, 1 refrigerator, one door towards the corridor, one door to the seminar room, the other door in seminar room towards the outside. Human activities and the working machines are unstable factors in the experiment. Air condition working time is from 8:00 to 20:00 in the working day. The temperature was automatically controlled by a sensor in laboratory.

The simulation experiment aims to examine existence of obviously effect of the different number and location of humidifiers. The experiment was divided into 4 periods of one week each. Small size sensor used to record humidity and temperature. Two humidifiers were set up in laboratory. In order to verify the humidifier effectiveness, sensors were set up in different distance from the humidifier. This experiment lasted for four weeks which each week had a different experiment period. As the reference, there was no humidifier set up in the first week. Humidifier set up was starting from the second week to the last week. The total numbers and position of humidifiers changed every week. The placement of measurement point is shown in the Fig.2-12

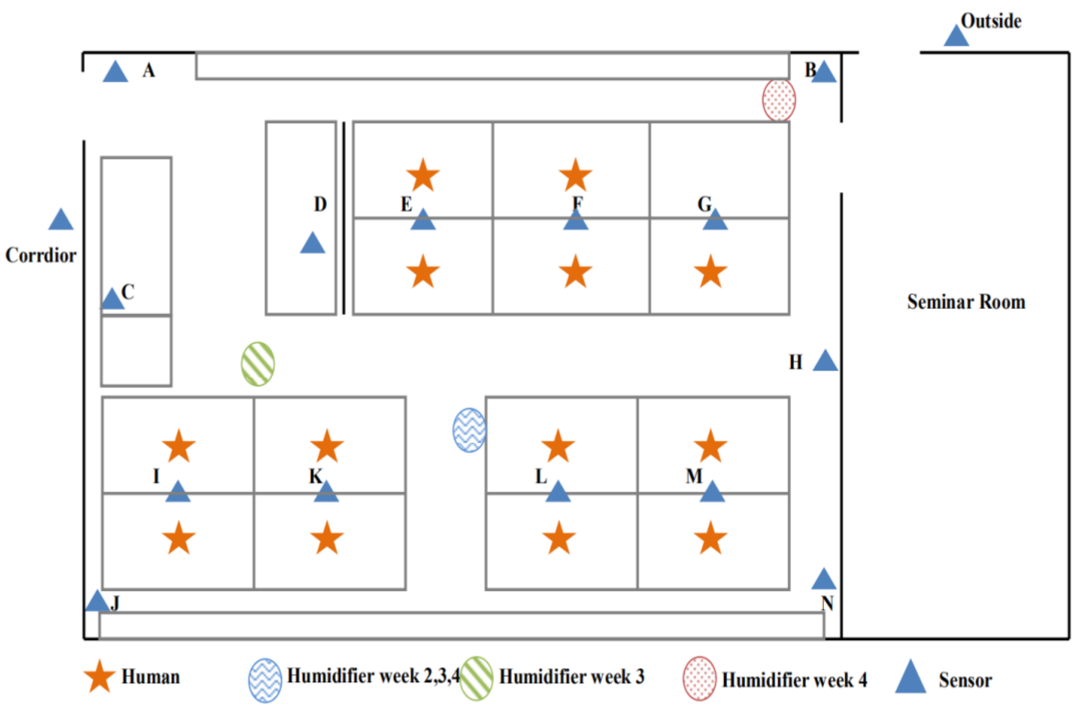
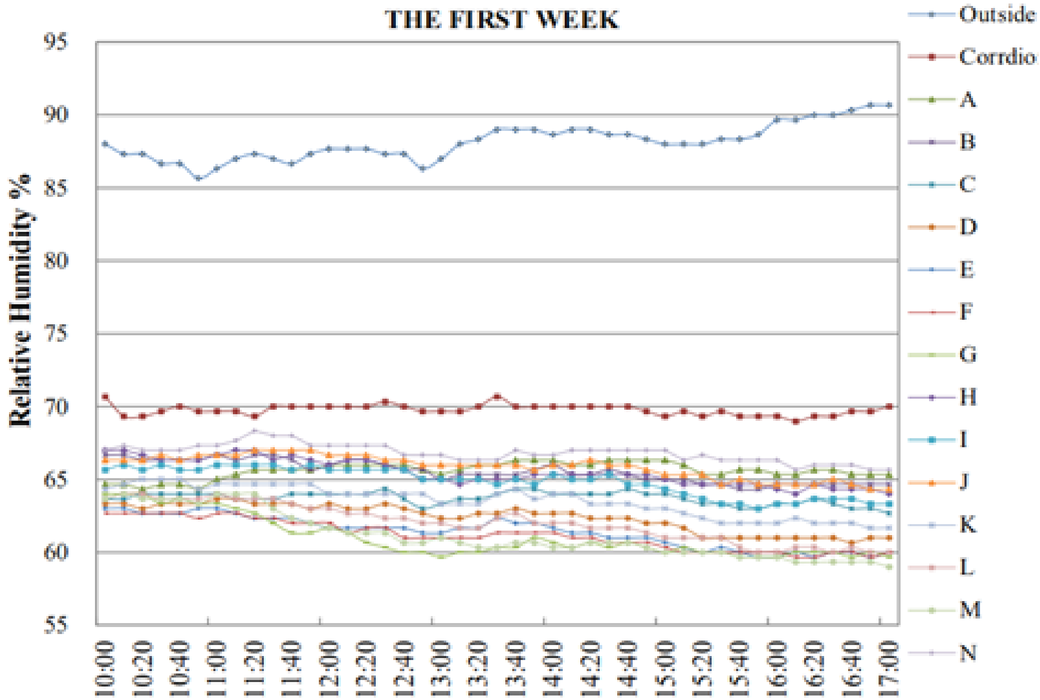
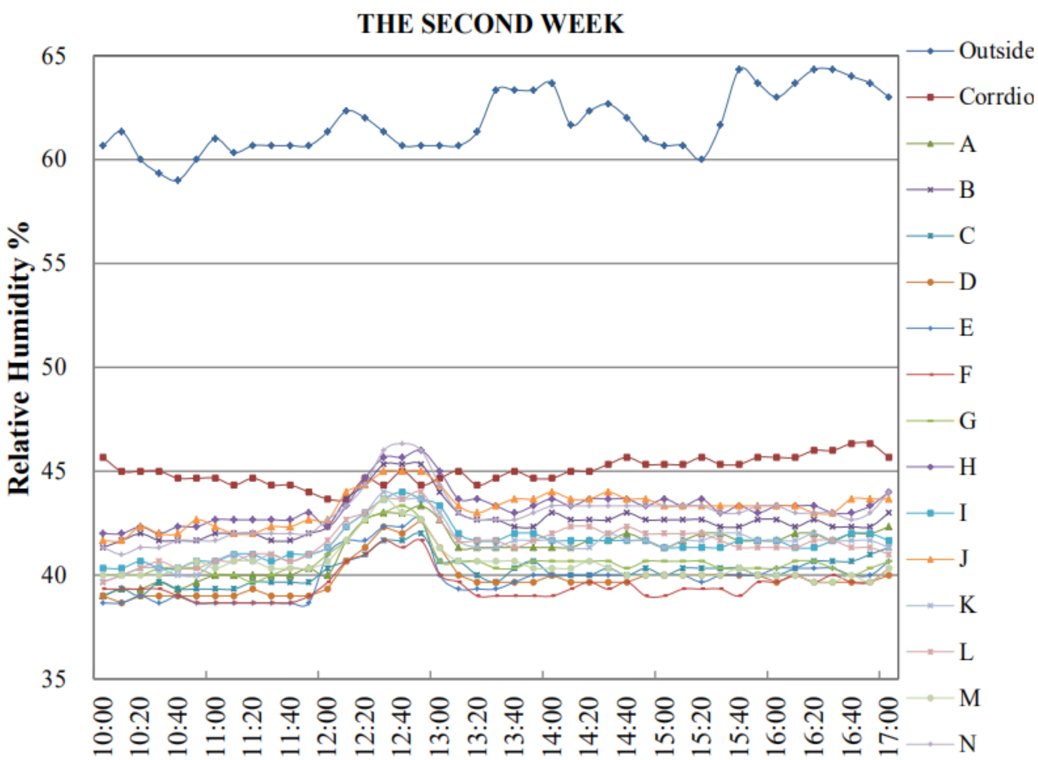



Figure 2-12 The placement of measurement point

In this experiment, result shows that the indoor humidity condition is mainly affected by outdoor humidity and temperature. Indoor humidity and corridor humidity changed with outdoor humidity. In the same room, humidity is higher near the wall area. Corridor test point’s humidity was obviously higher than indoor test points. The influence of ventilation condition to humidity is also very important.

Temperature condition and humidity condition of all of points are the opposite. Temperature of high humidity test point is on the low condition. Although the indoor humidity was mainly influenced by outdoor thermal environment, the use of humidifier affects the nearby area. Within the scope of the nearby, the air just can hold limited capacity water vapor. The humidifier working efficiency was infected by the different position of humidifiers. (Fig.2-13)



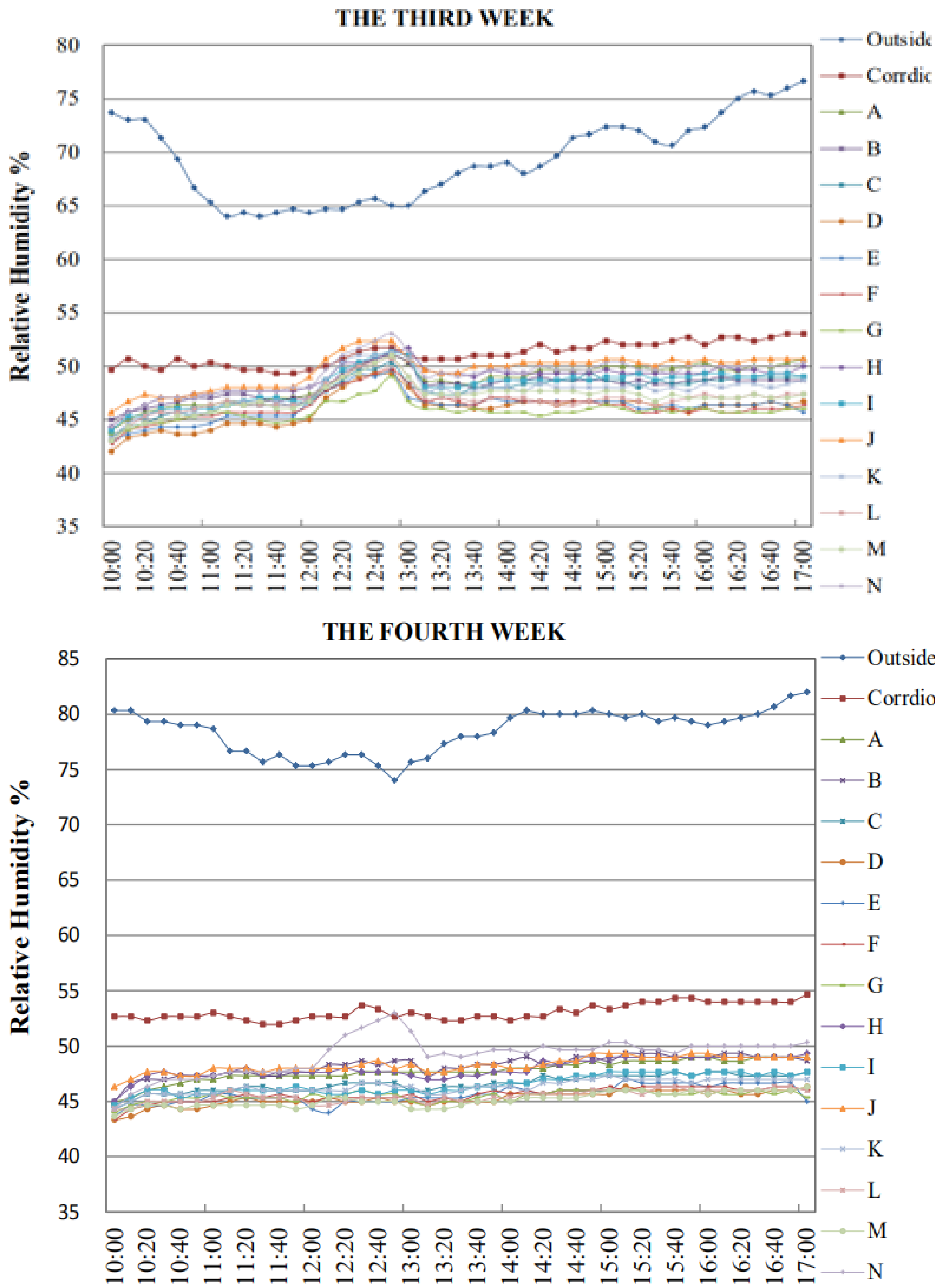



Figure 2-13 Average relative humidity during experiment time

In the building indoor environment, generally the water vapor in the humid air is in a superheated state and the content is very low, so at a certain extent, it can be approximated as the ideal gas. At the same time the density is small, the diffusion of wet air can be ignored.

The basic laws of conservation of heat and mass transfer between the fluid and the fluid and between the fluid and the solid interface are: mass conservation law, momentum conservation law, and energy conservation law. The mathematical models that generally describe these conservation laws are the corresponding governing equations, namely the mass continuity equation, the momentum equation, and the energy equation.

As follows：
Mass continuity equation：

$$\frac{\partial \rho}{\partial_t} + \nabla \cdot (\rho U) = 0 \tag{1}$$

In the formula, ρ is the fluid density, kg/$m^3$
U is the gas velocity vector, $U_{x,y,z}$.
In this study, considering air as an incompressible fluid, the continuity equation can be simplified to:

$$\nabla U = 0 \tag{2}$$

Momentum equation:

$$\frac{\partial(\rho U)}{\partial_t} + \nabla \cdot (\rho U \otimes \mathrm{U}) = -\nabla P + \nabla \cdot \tau + \rho_g \tag{3}$$

where ⊗ is the outer product: $U \otimes \mathrm{V} = \mathrm{U}V^T$,

$\rho$ is the fluid density,

U is the flow velocity vector,

∇ ·is the divergence,
P is the pressure,
t is the time

$\tau$ is the derivatoric stress tensor, which has order two,$\tau = \mu(\nabla U + (\nabla U)^T) - \frac{2}{3}\delta\nabla \cdot U$

g represents body accelerations acting on the continuum, for example gravity, inertial accelerations, electrostatic accelerations.
The energy equation：

$$\frac{\partial \rho h_{tot}}{\partial t} - \frac{\partial_p}{\partial t} + \nabla \cdot (\rho U h_{tot}) = \nabla \cdot (\lambda \nabla T) + \nabla \cdot (U \cdot \tau) + U \cdot S_M + S_E \tag{4}$$

In the building indoor environment, generally the water vapor in the humid air is in a superheated state and the content is very low, so at a certain extent, it can be approximated as the ideal gas. At the same time the density is small, the diffusion of wet air can be ignored.
For a fixed flow field, there is no indoor emission source and the wall is adiabatic, the initial concentration is zero. When a certain outlet ($\mathrm{n}_s$) from the $0^{\mathrm{th}}$ time constant release of a certain concentration (C), The air transient accessibility at any arbitrary point (P), at any time ($\tau$) in the space is defined as follows:

$$a_s^{n_{s,p}}(\tau) = \frac{d^P(\tau)}{d^{n_s}} \tag{5}$$

Where: $d^{n_s}$ is the $n_s$th air outlet air moisture content;

For a fixed flow field, there is no air conditioning and the wall is adiabatic, the initial concentration of each inlet concentration and rooms were 0. When a certain wet source ($n_c$) from the 0th time constant release of a certain distribution intensity $J_d^{n_c}$,the wet source transient accessibility at any arbitrary point (P), at any time ($\tau$) in the space is defined as follows:

$$a_c^{n_{c,p}}(\tau) = \frac{d^P(\tau)}{d_E^{n_c}}, d_E^{n_c} = \frac{J_d^{n_c}}{Q} \tag{6}$$

Where: $J_d^{n_c}$ is the distribution intensity of $n_c$th wet source ;( g/s)

$d_E^{n_c}$ is the average concentration of the inlet at steady state ; (g/kg)

Q is the room ventilation mass flow. ($\rho \times kg$/s)

For a fixed flow field, there is no wet source and the wall is adiabatic, each airflow inlet concentration is 0. At the 0th time there is an initial distribution of wet air in the room, then at any time thereafter (τ), the transient accessibility of initial condition (TAIC), at any arbitrary point(p) is defined as follows:

$$a_I^p(\tau) = \frac{d^P(\tau)}{\overline{d_0}}, \overline{d_0} = \frac{\oint d^P(0)dV}{V} \tag{7}$$

Where: $\overline{d_0}$ is the average of the moisture content of the air in the room at the 0th time

Transient accessibility series concept is determined as dimensionless number. Reflecting the various air outlets, indoor humidity source and the indoor air initial condition in the room instantaneous moisture content of any point of the degree of impact. These concepts are the time-related indicator, the value of the room only by the air organization and has nothing to do with the air supply concentration. The transient accessibility of supply air reflects the inherent properties of the flow field. Its value can be obtained by actual measurement or simulation of indoor air moisture content 0 .

Therefore, according to the principle of linear superposition, when the flow field conditions determined, can be obtained in a variety of fixed boundary conditions at the same time. The instantaneous moisture content d(g/kg) of any point (p) at time(τ) can be expressed as:

$$d^P(\tau) = \sum_{n_s=1}^{N_s}\left[d^{n_s} a_s^{n_{s,p}}(\tau)\right] + \sum_{n_c=1}^{N_c}\left[\frac{J_d^{n_c}}{Q}\ a_c^{n_{c,p}}(\tau)\right] + \overline{d_0}\, a_I^p(\tau) \tag{8}$$

When the state is stable, the contribution of the initial condition is 0, and the indoor humidity distribution is affected by the source, source and boundary condition formula is shown in (5).

$$d^P(\infty) = \sum_{n_s=1}^{N_s}\left[d^{n_s} a_s^{n_{s,p}}(\infty)\right] + \sum_{n_c=1}^{N_c}\left[\frac{J_d^{n_c}}{Q}\ a_c^{n_{c,p}}(\infty)\right]$$

(5) First, In the formula “d” is the absolute humidity. The absolute humidity between the mass of water vapor present in moist air to the mass of dry air. Absolute humidity is normally expressed in kilograms of water vapor per kilogram of dry air.

Absolute humidity expressed by mass:

$$\mathrm{d} = \frac{\mathrm{m_w}}{\mathrm{m_a}} \tag{9}$$

Where: d = absolute humidity; $\mathrm{m_w}$= mass of water vapor (kg): $\mathrm{m_a}$= mass of dry air (kg)

Based on the experiment of Lu Xin, the ANSYS FLUENT and AIRPAK series software was used to simulate the indoor humid environment and humidifier conditions. The temperature and humidity data of the experiment used as external boundary condition. According to the actual construction of the laboratory and indoor equipment distribution, the physical model of numerical simulation was established as shown in Fig 3 The room has a geometric size of 8m* 6m * 2.8m. The airflow form is the upper air supply and the return air. Totally two (0.6 * 0.6) outlets and four (0.6 * 0.6) return air inlets are set on the ceiling. The room has a total of four rows of 4 lamps (1.5m* 0.2m* 0.2m), 14 people, 15 computers and 2 printers. In the process of simplification, only objects with less impact on the airflow in the laboratory are simplified, and most of the space and the actual situation of laboratory are preserved. And the main simplified parts are followed. The computer, the printer and the fluorescent lamp are replaced by the block. The table only retains the desktop part, the table is adiabatic, and only as an obstacle. The door of the room is simplified for the temperature boundary entrance. The application of CFD technology in indoor thermal environment is based on the discretization and numerical analysis of the mass, momentum and energy conservation differential equations of incompressible gas in the room.

From the molecular motion theory, the free travel of gas molecules is 1*10-11mm. The value and the room length L compared to the specific value of the higher order, that is 1＜L<< 1， it can be considered as Museum. The inner air is a continuous medium.

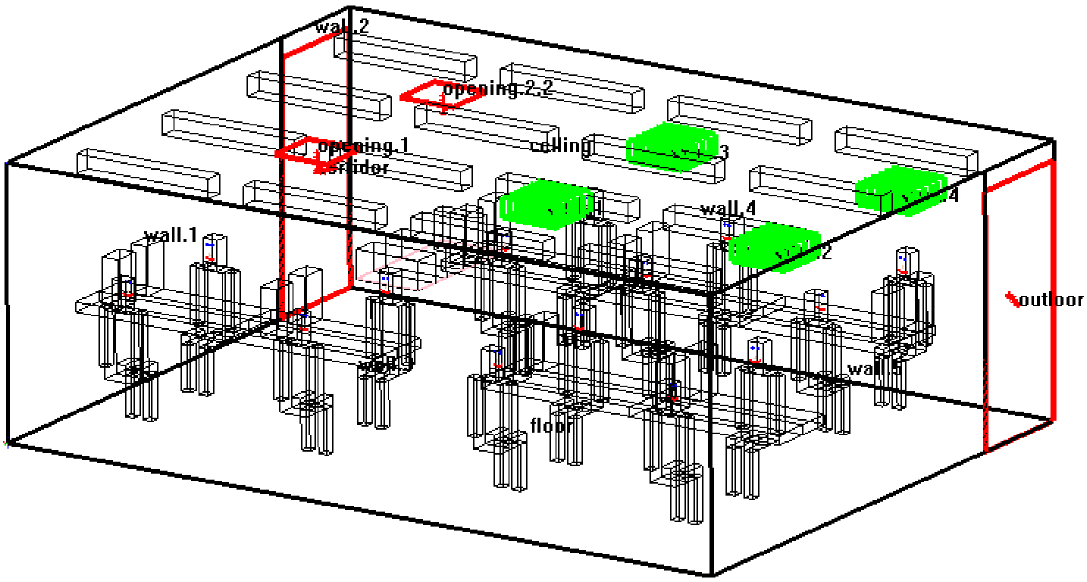



Figure 2-14  The physical model layout in 3D view

The flow that flow with time changed are called unsteady flows; flow of time does not change with the flow of time known as steady flow. In the actual situation, when the air flows through the air outlet at a uniform rate into the room, after a period, the air supply air and indoor air fully mixed, the indoor flow field in a steady state, this time the indoor flow field can be regarded as Steady flow

According to the knowledge of fluid mechanics, if the flow rate is sufficiently small compared with the speed of sound and the Mach number $M \leq 0.25$, the compressibility of the gas is negligible, the air supply wind speed is generally within 15m / s, the wind speed is much lower than the sound speed. In this velocity range, the pressure and temperature in the flow field are smaller than those of the free flow, so the corresponding density change can be neglected, and it can be considered that the air is incompressible. The density is assumed as boussinesq. That is the fluid density and the dependence on the ambient temperature changes, regardless of pressure. Based on the microscopic analysis of the air turbulence characteristics, the indoor zero equation model was used in this simulation and the control equations was discretized with the control volume dispersion method, considering the effect of radiation heat transfer. The set of relevant parameters of discrete solution is shown in Table 2-2

Table 2-1 The set of relevant parameters of discrete solution

| Turbulence model | Varlables | Pressure | Temp. | Momenturn | Velocity | Water vapor | Body force |
|---|---|---|---|---|---|---|---|
| The indoor zero equation | Discertization schemes | Second order upwind | | | | | |
| Conergence criterion | Relaxation factor | 0.7 | 1.0 | 0.3 | 1.0 | 0.1 | 0.1 |
| Flow 0.001 Energy 0.0000001 | Solver schemes | Pressure AMG | Temp. AMG | Momenturn AMG | Kinetic energy | Kinetic energy Dissipation rating | Water vapor |
| | Circulation type | V | Flex | | | | |

Table 2-2 The calculation condition

| Thermal Specification | | Physical model | | | | | | | |
|---|---|---|---|---|---|---|---|---|---|
| | | air condition 1 | air condition 2 | Fan 1 | Fan 2 | Fan 3 | Fan 4 | Outdoor | Corridor |
| Temp.(℃) | case 1 | 21.5 | 21.9 | 21.8 | 21.7 | 21.5 | 21.6 | 18.4 | 21.7 |
| | case 2 | 20.3 | 20.0 | 20.3 | 20.0 | 20.0 | 19.9 | 13.7 | 18.6 |
| | case 3 | 21.9 | 21.7 | 21.9 | 21.7 | 21.5 | 21.5 | 13.7 | 19.2 |
| Species(RH). | case 1 | 41.2 | 41.4 | 41.2 | 41.7 | 41.0 | 43.0 | 88.0 | 68.0 |
| | case 2 | 46.1 | 46.8 | 45.9 | 46.8 | 45.5 | 47.9 | 68.0 | 49.0 |
| | case 3 | 45.3 | 45.6 | 45.0 | 45.5 | 45.3 | 46.8 | 78.0 | 54.0 |
| Velocity(m/s) | case 1 | 0.2 | 0.2 | 0.2 | 0.2 | 0.1 | 0.1 | 0.1 | 0.1 |
| | case 2 | 0.2 | 0.2 | 0.2 | 0.2 | 0.1 | 0.1 | 0.1 | 0.1 |
| | case 3 | 0.2 | 0.2 | 0.1 | 0.1 | 0.1 | 0.1 | 0.0 | 0.0 |
| Air Volume | case 1 | 540.0 | 600.0 | 540.0 | 550.0 | 428.0 | 480.0 | 450.0 | 450.0 |
| | case 2 | 540.0 | 600.0 | 540.0 | 550.0 | 428.0 | 480.0 | 450.0 | 450.0 |
| | case 3 | 540.0 | 600.0 | 440.0 | 450.0 | 328.0 | 380.0 | 250.0 | 250.0 |

In this CFD simulation, the wall structure is insulated. The body's heat and moisture volume were set as adult man 22 ℃. Human body model for sitting, heat 50w / m2 clothing thermal conductivity of 0.5clo (0.078m2/ k*w). The calculation condition parameters are shown in Table 2-12.

There are two humidifiers in the experiment. A humidifier was set at the room center in Case 1. And two humidifiers were set in the Case 2. In Case 3, the position of humidifiers has been changed. Fig.2-15 shows the position of humidifiers in each case. The humidifier position marked with orange color. Humidifiers parameters are as followed. Humidifier 1: Total power is 40W, Species: Air 0.008g/s, Water 0.075g/s. Humidifier 2 for case 2 and case 3: Total power is 40W Species: Air 0.005g/s, Water 0.05g/s.

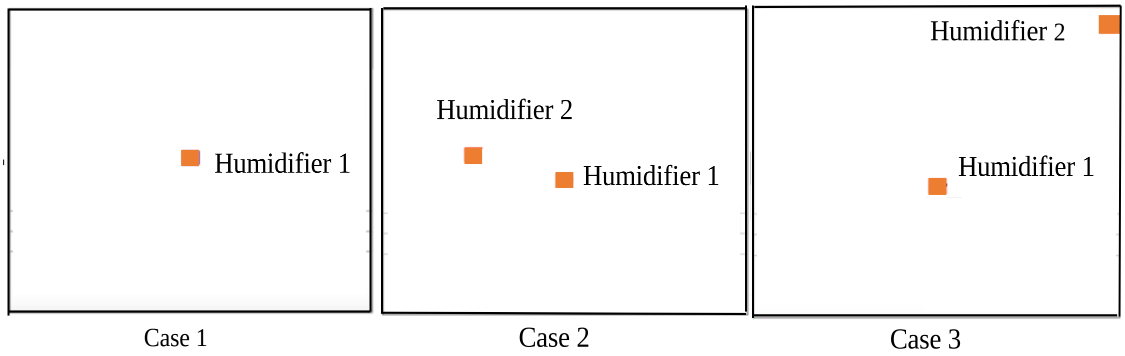
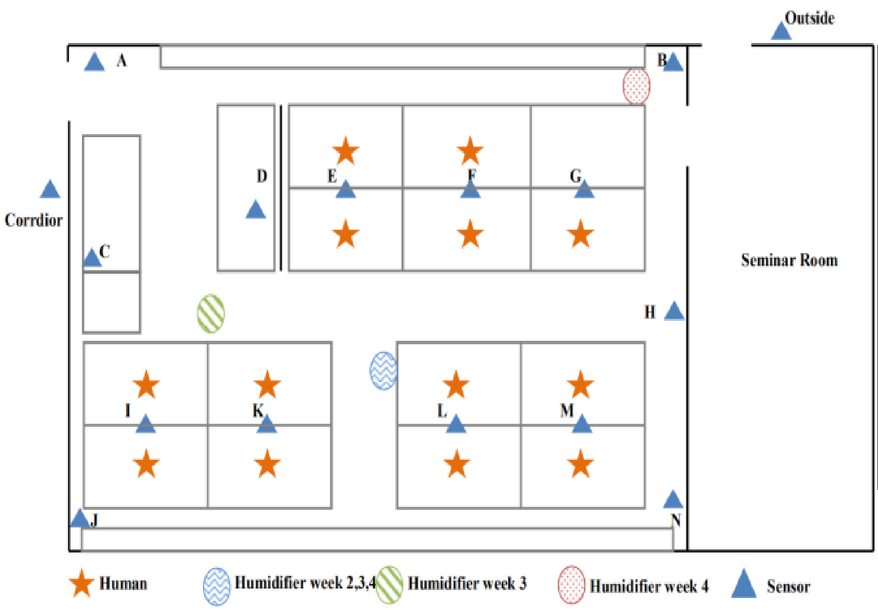



Figure 2-15 The position of humidifiers

In the simulation, 7 major points were selected. The placement of measurement points is shown in the Fig.2-16. The Fig2-17 shows the comparison result between the measurement and simulation. The relative humidity difference is small and the error range does not exceed 5%. The location near the air inlets and outlets, at the experiment period affected by the airflow organization,so that the error of temperature slightly large.

And the person has a different body heat. But in the simulation the body's heat and moisture volume were set as the adult man heat body 22 ℃, heat 50w / m2 clothing thermal conductivity of 0.5clo (0.078 m2/ k * w) so that the error of temperature slightly large.

Overall, the results of simulated experiment agree well with the measured results, which proves the effectiveness of the model. In summary, it can be certified that the model has a good feasibility. It can be realistic response velocity, temperature and humidity distribution. The error range remains within acceptable limits.




| Point | Position |
|---|---|
| A | (0.9,1.6) |
| B | (2.6,1.6) |
| C | (5.5,1.6) |
| D | (7.2,1.6) |
| E | (3.1,4.6) |
| F | (4.8,4.6) |
| G | (6.5,4.6) |

Figure 2-16 The placement of measurement point

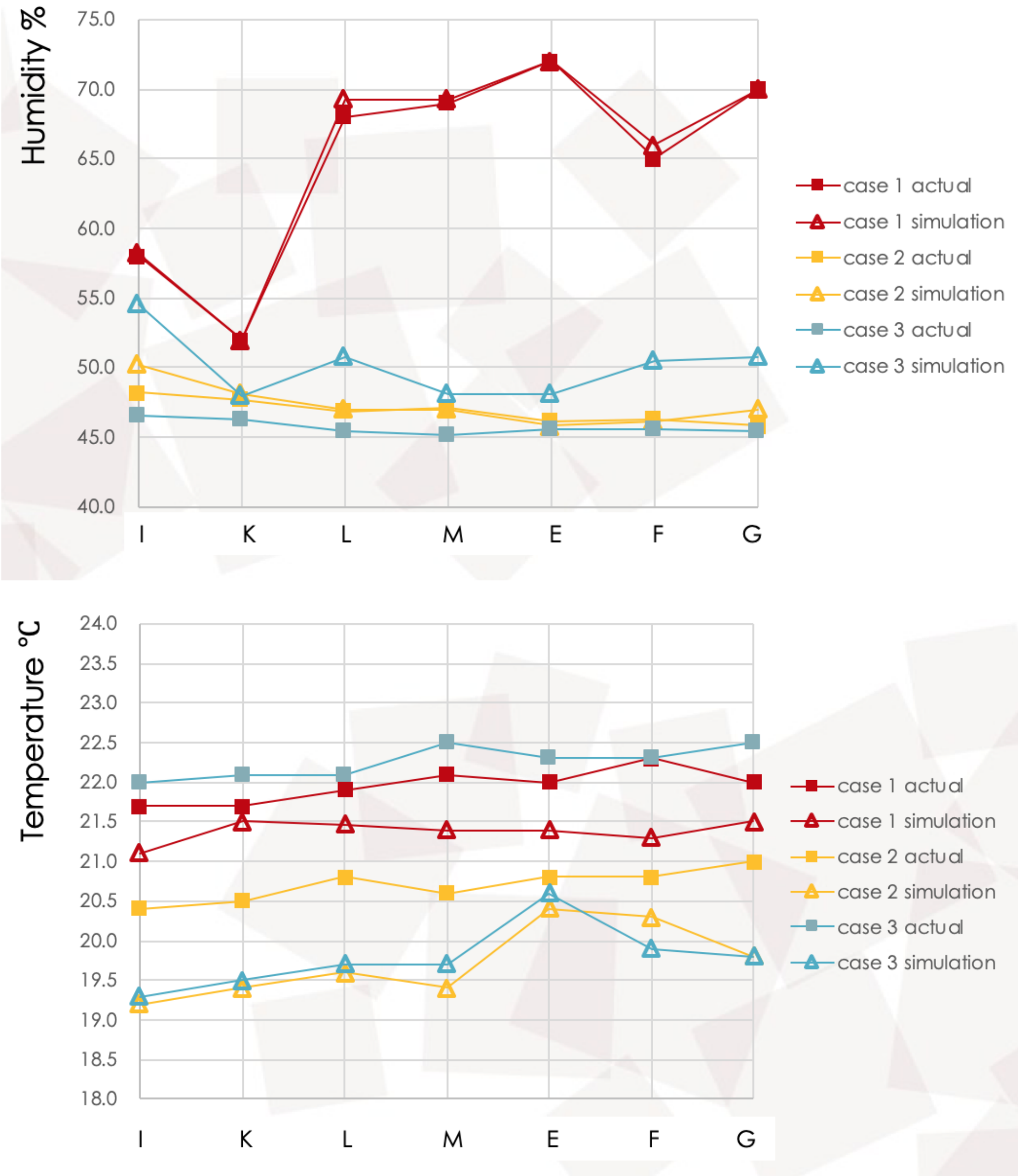



Figure 2-17  Comparison between measured and simulated point

# CHAPTER 3
# THE EFFECT OF DIFFERENT VENTILATION METHODS ON INDOOR HUMIDITY BY CFD SIMULATION METHOD

## 3. The Effect of Different Ventilation Methods on Indoor Humidity by CFD Simulation Method

Nowadays, people spend most of their time (70% -90%) indoors, and indoor air quality (IAQ) directly affects people's production, life and health, so IAQ is attracting more and more attention. With the development of air conditioning technology, research into the indoor air environment is already shifting from comfort to healthcare air conditioning. The requirements for human comfort are met by air-conditioning systems that regulate indoor environmental parameters or exploit the potential of natural energy sources. In particular, telecommuting has increased due to the COVID-19 pandemic of late, and this trend is expected to continue for a while. The ability of the virus to survive in the air depends on environmental conditions, especially ambient temperature and relative humidity are the most important parameters. A series of studies have shown that the dry environment seems to create an excellent environment for the growth and spread of influenza viruses.

Since the 1940s, many scholars around the world have studied the characteristics of jets in mechanically ventilated rooms, both experimentally and theoretically, and over the course of a decade, a series of jet formulas were established and used to predict indoor air distribution, which became the simplest method of predicting indoor air distribution at the time. The Zonal model was formally proposed in 1970, stemming from a study of the analysis of temperature stratification in heated rooms. Since then, researchers have used the model to study various forms of rooms and airflow patterns to predict the amount of natural ventilation and temperature distribution in a room, and at that time the model was proposed to be restricted to natural ventilation, but later it was suggested that the method could also be used for mechanical ventilation based on the results of the study. CFD has the unique advantages of low cost, fast speed, complete data and can simulate various working conditions. The future is favored by people. Since 1974, when CFD techniques were first applied to the simulation of air flow in ventilated rooms by Nielsen in Denmark, the research and application of CFD in HVAC has developed both rapidly and well.and CFD methods have been continuously developed since then. Markatos carried out a CFD numerical simulation of a large television broadcast room to derive three-dimensional indoor flow and heat transfer characteristics to provide a basis for improving the design of air conditioning in large spaces.Lee and Kisup used CFD to discuss the effects of different factors on the air conditioning effectiveness of split-level air conditioning systems, such as air outlet location and space type, respectively.A comparison of the energy efficiency and temperature field of different air supply for a large stadium was carried out by Hua Ya Wei,who concluded that the downward and upward air supply is better than the upward and downward air supply ;The main factors affecting the indoor temperature field and temperature drop in the side and bottom return air supply methods were identified by Danny Wu;Yang Li conducted numerical simulations of three air supply angles for indoor cabinet air conditioners and obtained the characteristics of the respective flow fields and temperature distribution; Tang Qiang simulated the temperature field of three different air velocities under a fixed air supply method and showed that the air supply velocity affects the indoor thermal environment; Liu Caixia used CFD software to simulate the change of airflow organization in an air-conditioned office with the door open and closed to show that the airflow organization affects the thermal environment of the indoor area. Qu Yunxia discusses the numerical simulation analysis of the indoor thermal environment under four air supply methods.. Di Yuhui, Wang Shancong used airpak simulations to analyse errors in indoor airflow organisation. Jiang Lvlin, Wang Changling studied the numerical simulation of indoor natural ventilation environment. Cairui Yu optimally simulated the thermal comfort of a classroom based on Airpak.Zhang Yameng analyzed the ventilation airflow organization of an air-conditioned office based on CFD. In previous research, CFD simulation techniques have been used to numerically simulate various types of large spaces and various types of airflow organization, and the simulation results can provide a basis for improving the thermal environment and airflow organization in large spaces.

Humidity is an important parameter for evaluating indoor air quality, which not only affects the thermal comfort of the human body but also seriously restricts the function of the building.. In terms of the comfort of the human living environment, a high humidity environment will promote the propagation of mold and the growth of pathogens, which in turn induces symptoms such as asthma and allergies, which seriously threatens the health of the occupants. Discomfort in the respiratory tract, and even lead to the generation of static electricity, causing potential safety hazards. For some special buildings, such as operating rooms, intensive care units, swimming pools and plant greenhouses, temperature and humidity control are critical. In addition, dew condensation on walls, floors and roofs is also a serious problem faced by buildings. When the moisture content of the air near the walls exceeds the corresponding saturation moisture content of the wall temperature, it will cause dew condensation on the walls, and then It leads to peeling of the wall surface, mildew, corrosion of the enclosure structure and reduction of the life of the building. At present, two methods are often used in the engineering field to reduce the condensation risk of the enclosure structure: ordinary rooms use walls to attach insulation materials to increase the dew point temperature of the inner surface of the enclosure structure; Ventilation and air conditioning rooms reduce the possibility of condensation of wet air on the wall by controlling the temperature of the room and the humidity of the indoor air. Compared with the two methods, the use of mechanical ventilation can not only reduce the risk of condensation in the building, but also regulate the indoor temperature and humidity environment, so ventilation is more widely used in engineering. Because the indoor humidity distribution of the building is related to the moisture production characteristics of the source, and it changes with the change of the building ventilation conditions. Therefore, by studying the influence of ventilation on the distribution of indoor wall condensation, it has certain reference value for the theoretical research of building moisture resistance and condensation prevntion and the design of ventilation engineering. For compact spaces, such as offices, the impact of different air supply methods on the indoor thermal environment has been less studied. CFD simulation techniques can be used to simulate the indoor thermal environment created by air conditioning in small spaces, revealing the impact of different air supply speeds and different air supply angles on the indoor thermal environment under fixed air supply methods. This paper uses Airpak software to numerically simulate three common air supply methods in offices, namely top air supply and return, side down air supply and side up air return, and side up air supply and side down air return, and to compare and analyze the simulation results in order to select the most suitable airflow organization for the air supply method and provide a methodological and theoretical basis for improving the thermal environment design of air-conditioned rooms. Thermal comfort is the condition of mind that expresses satisfaction with the thermal environment and is assessed by subjective evaluation (ANSI/ASHRAE Standard 55). Maintaining this standard of thermal comfort for occupants of buildings or other enclosures is one of the important goals of HVAC (heating, ventilation, and air conditioning) design engineers. Age of air is the age of air mass (Age of air), refers to the air mass since the entry into the room to a point in the room experienced time, reflecting the freshness of indoor air, it can be a comprehensive measure of the room ventilation effect, is an important indicator to evaluate the quality of indoor air. The concept of air age was first introduced by Sandberg in the 1980s. By definition, air age refers to the time when air enters a room. When pollutants are evenly distributed in a room and the air supply is fully fresh, the smaller the air age at a point, the fresher the air at that point and the better the air quality. It also reflects the room's ability to remove pollutants, with a room with a small average air age being more capable of removing pollutants. Due to the obvious physical significance of air age, it is widely used as an important indicator of the freshness of air in air-conditioned rooms and the ability to change air. Traditionally the air age concept has only considered the interior of the room, the air age at the room inlet is considered to be 0 (100% fresh air). In order to consider the effect of the entire ventilation system including return air, mixed air and flow in the ducts, Tsinghua University has proposed the concept of full air age The time that the air micro-clusters experience since entering the ventilation system; the air age at the entrance of the room is taken as 0 and the resulting air age is called the room air age. Compared to the room air age, the full air age can be seen as an absolute parameter and the full air ages of different rooms can be compared.

## 3.1. Methods

Airpak was developed and produced by Fluent in the USA and is now widely used to simulate indoor and airflow organization distribution. Airpak has many advantages: it can automatically perform meshing and check the sparseness, length to thinness ratio and quality of the mesh, and can locally encrypt specific parts of the mesh; it has powerful post-processing functions and comprehensive numerical reports, and a single calculation can provide Images and data results of indoor air quality technical indicators such as indoor airflow organization, PMV, PPD and other measures of indoor air quality; the use of Airpak allows design risks to be predicted, thus reducing design costs. Airpak 3.0 software is used for this project, based on CFD simulation methods and using standard k-models. (1) Collecting data and ensuring its accuracy. (2) Establishing the model required for airflow organization analysis according to the preliminary data and the requirements of the analysis software.(3) Set reasonable control equations, initial conditions and boundary conditions, and divide the mesh and solve the equations.(4) Simulate and analyze the velocity, temperature and humidity fields of the airflow organization with the help of the software.(5) Based on the results of the airflow analysis, assess whether the air conditioning and ventilation solutions for the room or area are reasonable. The operation flow of the air flow simulation is shown in Figure 3-1.

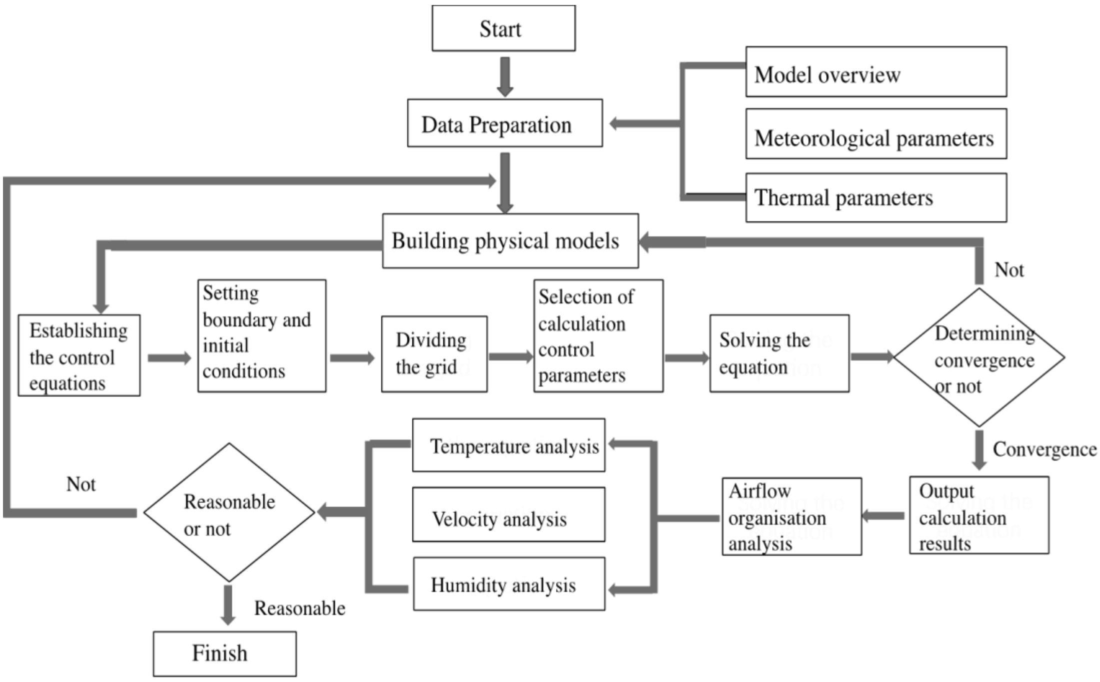



Figure 3-1 The operation flow of the air flow simulation

## 3.2. The summary of CFD simulation

### 3.2.1. Establishment of a basic physical model

Air conditioning system is the simulation method for the verification of the cases. And the model for establishing the simulation shown in Figure 1 is the research model. The model is equipped with air inlet and outlet with three different types, the sizes of the simulated actual model established are 3.5 m × 3 m × 2.6 m (length × width × height). As for the thickness on the surface (wall) and under the ground, it does not need to be considered, because it does not have any influence on the flow field. In this study to investigate the effect of the air supply and air outlets position on the indoor environment and humidity distribution, three different combinations of air supply and outlets were set up and simulated by various combinations of positions. Through the combination of various opening positions, the simulation is made to compare the changes of different opening positions to the internal humidity of the model and to evaluate the advantages and disadvantages of natural ventilation convection. The selected analysis space has three different changes. This paper simulated the effects of three different ventilation methods and air supply parameters on condensation in four cases. The Case 1 is the upper supply and back mode, the Case 2 is the upper supply and lower back mode, and the Case 3 condition is the lower supply and upper back mode. The wall in Case 4 are adiabatic and dehumidified. The wall of Case 1, Case 2 and Case3 is made by concrete with overall heat transfer coefficient of 1,86W/m2・K. The physical model is summarized in Fig.3-2. The model boundary conditions are set in the Table 3-1, the parameters setting of simulation calculation condition are shown in Table 3-2.

### 3.2.2. Simplification of the physical model

If the simulated state following the actual state of lab, the error between the actual measurement and the simulation experiment will be minimized. However, due to the accuracy of the model will affect the division of the mesh, the operation of the computer's performance requirements is high, It is difficult for ordinary computers to meet the requirements, so in order to simulate the smooth progress of this model I affect the room airflow organization smaller objects to a certain degree of simplification, thereby reducing the operation of the computer performance requirements. Main simplified part

5 points simplifying assumption were made for the model：

1.The effects of gravity fields on air and water vapors are ignored;

2.Both air and water vapor are considered to be incompressible fluids with a constant density;

3.When the water vapor evaporates from the water surface, only latent heat exchange is performed, and sensible heat exchange is not considered;

4.When water vapor condenses on the wall, the heat release ignored from the phase change process;

5.Wall surfaces moisture absorption capacity is not considered during wall condensation.

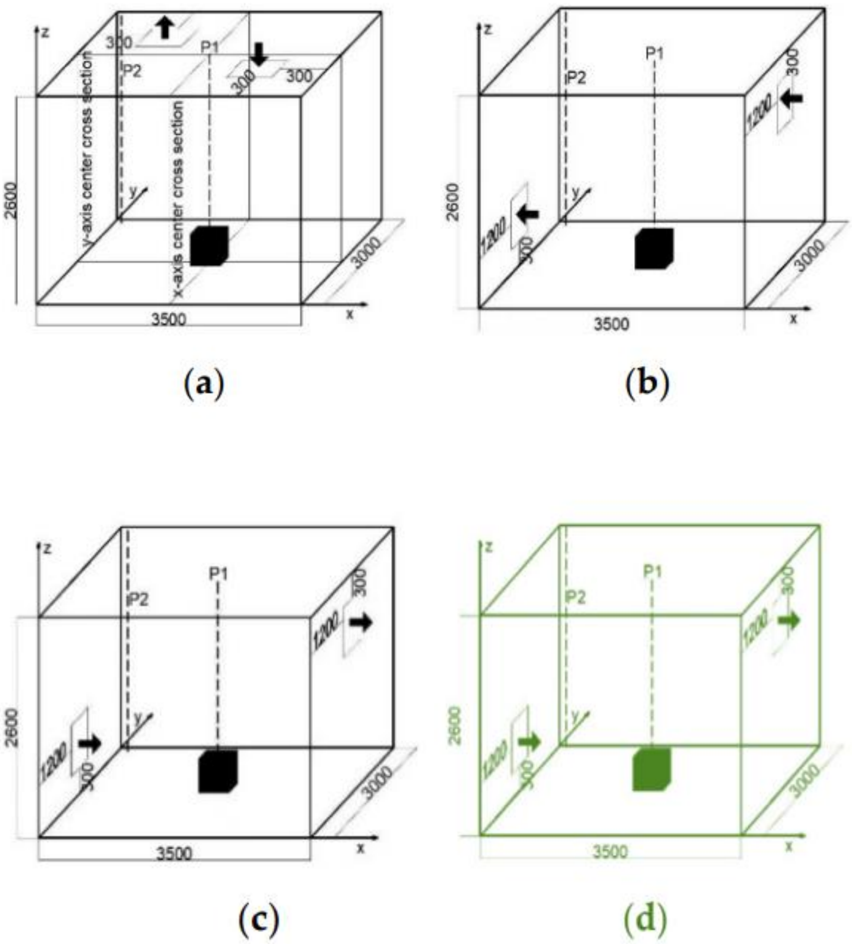



Figure 3-2 The physical model of each ventilation condition

Table 3-1 Model Boundary Conditions.

| **Boundary** | **Parameter Setting** |
|---|---|
| Air supply | Air volume :216m^3/h, Temp: 29 °C, Relative Humidity: 40% |
| Air outlet | Outflow |
| Wall | Dehumidification, overall heat transfer coefficient 1.86W/m2・K. Outdoor temperature: 5°C |
| Initial environment | Temp: 22 °C, RH: 60% |
| Humidifier | Amount of moisture 300ml/h, Air 0.008g/s Water 0.05g/s |
| Ceiling and Floor | Insulation |

Table 3-2 The simulation Calculation Condition

| **Case number** | **Outlet Arrangement** | **Ventilation volume (m^3/h)** | **Air supply parameter Temperature °C,** | **Humidity %** | **Humidity (g/s)** |
|---|---|---|---|---|---|
| 1 | Upper supply and mode | 216 | 29 | 40 | 0.05 |
| 2 | Upper supply and lower back mode | 216 | 29 | 40 | 0.05 |
| 3 | Lower supply and upper back mode | 216 | 29 | 40 | 0.05 |
| 4 | Lower supply and upper back mode | 216 | 29 | 60 | 0.08 |

### 3.2.3. Simplification of boundary conditions.

a) Continuous media. From the molecular motion theory, the free travel of gas molecules is 1*10-11mm, the value and the room length L compared to the specific value of the higher order, that is 1_L<< 1it can be considered Museum. The inner air is a continuous medium.
b) Steady flow field Flow of the flow of time with the flow of time known as the unsteady flow; flow of time does not change with the flow of time known as steady flow. In the actual situation, when the air flow through the air outlet at a uniform rate into the room, after a period of time, the air supply air and indoor air fully mixed, the indoor flow field in a steady state, this time the indoor flow field can be regarded as Steady flow.
c) Ideal incompressible air According to the knowledge of fluid mechanics, if the flow rate is sufficiently small compared with the speed of sound and the Mach number M ≤ 0.25, the compressibility of the gas is negligible, the air supply wind speed is generally within 15m / s, the wind speed is much lower than the sound speed. In this velocity range, the pressure and temperature in the flow field are smaller than those of the free flow, so the corresponding density change can be neglected, and it can be considered that the air is incompressible.
d) The density is assumed by boussinesq. The density is assumed by boussinesq. That is, the fluid density and the dependence on the ambient temperature changes, regardless of pressure.

### 3.2.4. Numerical simulation related parameters

The application of CFD technology in indoor thermal environment is based on the discretization and numerical analysis of the mass, momentum and energy conservation differential equations of incompressible gas in the room. Based on the microscopic analysis of the characteristics of air turbulence, this paper uses the indoor zero equation model and discretizes the control equations with the control volume dispersion method, taking into account the effect of radiation heat transfer. This study used FLUENT and Airpak company series professional software to study the indoor thermal environment simulation. Table 2-3 is the relevant parameters of discrete solution. Among the commonly used turbulence models, the k-ε model is the most widely used in engineering.

Therefore, the turbulence model in this numerical simulation will use the standard k-ε model.

From the molecular motion theory, the free travel of gas molecules is 1*10-11mm. The value and the room length L compared to the specific value of the higher order, that is 1＜L<< 1， it can be considered as Museum. The inner air is a continuous medium.

Table 3-3 The set of relevant parameters of discrete solution

| **Turbulence model** | **Varlables** | **Pressure** | **Temp** | **Momenturn** | **Velo city** | **Water vapor** | **Body force** |
|---|---|---|---|---|---|---|---|
| The indoor zero equation | Discretization schemes | Second order upwind | | | | | |
| Convergence criterion | Relaxation factor | 0.7 | 1.0 | 0.3 | 1.0 | 0.1 | 0.1 |
| Flow 0.001 Energy0.0000001 | Solver schemes | Pressure AMG | Temp AMP | Momenturn AMP | Kinetic energy | Kinetic Energy Dissipation rating | Water vapor |
| | Circulation type | V | | | Flex | | |

The flow that flow with time changed are called unsteady flows; flow of time does not change with the flow of time known as steady flow. In the actual situation, when the air flows through the air outlet at a uniform rate into the room, after a period, the air supply air and indoor air fully mixed, the indoor flow field in a steady state, this time the indoor flow field can be regarded as Steady flow. According to the knowledge of fluid mechanics, if the flow rate is sufficiently small compared with the speed of sound and the Mach number $M \le 0.25$, the compressibility of the gas is negligible, the air supply wind speed is generally within 15m / s, the wind speed is much lower than the sound speed. In this velocity range, the pressure and temperature in the flow field are smaller than those of the free flow, so the corresponding density change can be neglected, and it can be considered that the air is incompressible. The density is assumed as boussinesq. That is the fluid density and the dependence on the ambient temperature changes, regardless of pressure. Based on the microscopic analysis of the air turbulence characteristics, the indoor zero equation model was used in this simulation and the control equations was discretized with the control volume dispersion method, considering the effect of radiation heat transfer.

## 3.3. Mathematical method

In the building indoor environment, generally the water vapor in the humid air is in a superheated state and the content is very low, so at a certain extent, it can be approximated as the ideal gas. At the same time the density is small, the diffusion of wet air can be ignored.

The basic laws of conservation of heat and mass transfer between the fluid and the fluid and between the fluid and the solid interface are: mass conservation law, momentum conservation law, and energy conservation law. The mathematical models that generally describe these conservation laws are the corresponding governing equations, namely the mass continuity equation, the momentum equation, and the energy equation. As follows：

Mass continuity equation：

$$\frac{\partial\rho}{\partial_t} + \nabla\cdot(\rho U) = 0 \tag{3-1}$$

In the formula, ρ is the fluid density, kg/$m^3$； U is the gas velocity vector, $U_{x,y,z}$.

In this study, considering air as an incompressible fluid, the continuity equation can be simplified to:

$$\nabla U = 0 \tag{3-2}$$

Momentum equation:

$$\frac{\partial(\rho U)}{\partial_t} + \nabla\cdot(\rho U \otimes \mathrm{U}) = -\nabla P + \nabla\cdot\tau + \rho_g \tag{3-3}$$

where $\otimes$ is the outer product: $U \otimes \mathrm{V} = \mathrm{U}V^T$,$\rho$ is the fluid density ,U is the flow velocity vector,

$\nabla\cdot$is the divergence ,P is the pressure,t is the time$\tau$ is the derivatoric stress tensor, which has order two,$\tau = \mu(\nabla U + (\nabla U)^T) - \frac{2}{3}\delta\nabla\cdot U$

g represents body accelerations acting on the continuum, for example gravity, inertial accelerations, electrostatic accelerations.

The energy equation：

$$\frac{\partial\rho h_{tot}}{\partial t} - \frac{\partial_p}{\partial t} + \nabla\cdot(\rho U h_{tot}) = \nabla\cdot(\lambda\nabla T) + \nabla\cdot(U\cdot\tau) + U\cdot S_M + S_E \tag{3-4}$$

Where：

$h_{tot}$ is the total enthalpy of the fluid, $h_{tot} = h + 1/2U^2$ J/kg.

$h$ is the static enthalpy of the fluid, J/kg. And $\lambda$ is the thermal conductivity of the fluid, $W/(m\cdot K)$.

$S_E$ is the source of fluid energy.

There is no chemical reaction in the numerical simulation process of this subject, and the transport equation of the unreacted material is as shown in the following formula:

$$\frac{\partial(\rho Y_i)}{\partial_t} + \nabla\cdot(\rho v Y_i) = -\nabla J_i \tag{3-5}$$

$Y_i$ is the mass fraction of each substance predicted by FLUENT by the convective diffusion equation of the i$^{th}$ substance; $J_i$ is the diffusion flux of substance i, which is produced by the

concentration gradient?
When the moving fluid is laminar:

$$J_i = -\rho D_{i,m} \nabla Y_i \quad (3\text{-}6)$$

Where: $D_{i,m}$ is the diffusion coefficient of the first species in the mixture.
When the moving fluid is turbulent:

$$J_i = -\left\{\rho D_{i,m} + \frac{\mu_t}{Sc_t}\right\} \nabla Y_i \quad (3\text{-}7)$$

Where: $Sc_t$ is the turbulent Schmidt number
In the formula "d" is the absolute humidity. The absolute humidity between the mass of water vapor present in moist air to the mass of dry air. Absolute humidity is normally expressed in kilograms of water vapor per kilogram of dry air.
Absolute humidity expressed by mass:

$$\mathrm{d} = \frac{\mathrm{m_w}}{\mathrm{m_a}} \quad (3\text{-}8)$$

Where: d = absolute humidity; $\mathrm{m_w}$= mass of water vapor (kg):
$\mathrm{m_a}$= mass of dry air (kg)
The FLUENT 14.0 turbulence module (Standard k-ε) is used to analyze the flow field characteristics. The component transport module (Species Transport) and the multiphase flow module (Multiphase) simulated the mass transfer phase transition of water vapor in the air. The governing equation is as follows:

$$\frac{\partial(\rho\emptyset)}{\partial_\tau} + div(\rho u\emptyset) = div(\Gamma grad\phi) + s \quad (3\text{-}9)$$

Where: $\phi$is a general variable;vΓ is a generalized diffusion coefficient; s is a generalized source term.
Wall condensation conditions: When the absolute humidity $D_{air}$ of the air node around the wall surface is higher than the saturated absolute humidity $D_{wall}$ corresponding to the wall temperature, the wall surface is considered to be condensation.
Conditions for no condensation on the wall: Any time $\tau$ , when the absolute humidity $Dair$ of the air node around the wall is less than the saturated absolute humidity $Dwall$ corresponding to the wall temperature, the wall is not considered to be condensation

$$Dwall < Dair \quad (3\text{-}10)$$

### 3.3.1. Wet air enthalpy chart：

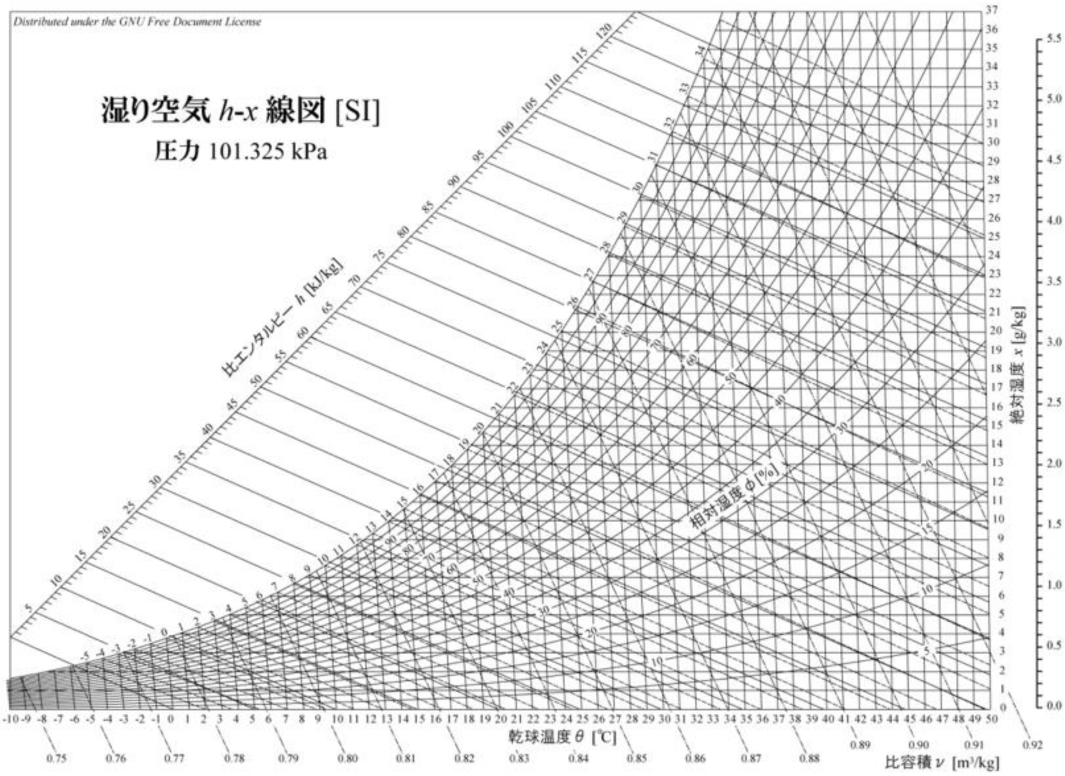



Figure 3-3 Wet air enthalpy diagram

Wet air enthalpy chart is referred to as a pneumatic diagram or humidity diagram. In the chart include dry bulb temperature, wet bulb temperature, absolute humidity, relative humidity, dew point temperature and enthalpy. And it can find all of parameters easily by setting any two values.

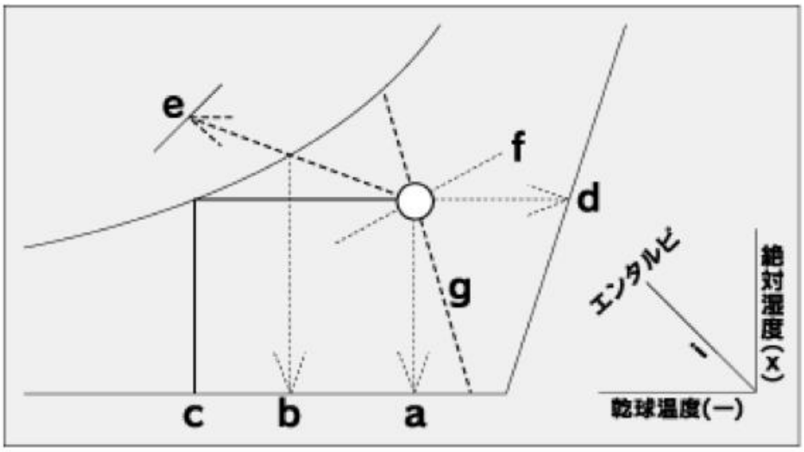



Figure 3-4 The explanation of wet air enthalpy diagram

And the wet air enthalpy diagram shows the relationship between the state of humid air, temperature and enthalpy. In the figure, the horizontal axis is temperature (a,b,c),the enthalpy on the slant side axis (e),and the absolute humidity on the vertical axis (d), and ‘f’ is relative humidity. When two of the state values of the humid air are determined, all other status values are found on the diagram.

### 3.4. Mesh generation and solution process

#### 3.4.1. Generate grid

In the numerical calculation of fluid flow and heat transfer, a very important step is to generate a grid, that is, to divide the continuous calculation area in space, divide it into many sub-areas, and determine the node. The final accuracy of the numerical calculation results of the flow and heat transfer problems and the efficiency of the calculation process mainly depend on the generated grid and the algorithm used.

Airpak 2.1 has automatic grid generation function, but the arrangement of grid parameters will directly affect the quality of the final grid. If the grid is too thick or too thin, it will affect the final simulation calculation results; If it is too thin, the huge memory consumption may prevent the computer from running normally; if the grid is divided too thin in some places and too coarse in other places, the calculation may not converge, which will ultimately affect the accuracy of the results.

Generally speaking, the grids used in the numerical calculation of fluid flow and heat transfer problems can be roughly divided into two categories: structured grids and unstructured grids. In general numerical calculation, the grids generated in orthogonal and non-orthogonal curvilinear coordinate systems are structured grids, which are characterized by the connection between each node and its neighbors being fixed and implicit in the generated In the grid, there is no need to specifically set up data to confirm this relationship between nodes and neighbors.

Unstructured grids have been rapidly developed since the 1980s due to their particular adaptability to irregular areas. In this grid, there are no fixed rules for the numbering of elements and nodes. The geometric information of the node must be stored, and the number of the cells adjacent to the cell must also be stored as the information of the connection relationship, so that the unstructured grid can store a large amount of information. High-quality meshes are the primary conditions for successful numerical simulation, and meshes that are too dense and too thin should be avoided. In order to ensure a high-quality grid, the discrete space of this subject uses a mixed hexahedral grid, and follows the following principles:

1. In places where the gradient is large, the grid must be sufficiently fine (temperature gradient, velocity gradient, etc.);
2. The size expansion ratio from one grid unit to another is generally less than or equal to 2;
3. Regular cube elements are best. Minimize the inclination of grid elements; long and flat elements should be avoided as much as possible.
4. Minimize the number of grids where the gradient is small, and distribute the grid reasonably where the gradient is large.
5. The calculation object can be divided into a uniform grid and a non-uniform grid, and a special detailed analysis (ie local encryption) can be performed on some objects in the calculation area and the surrounding grid to make the local The grid meets the computing requirements.
6. The number of grids between solid surfaces should not be less than 4.
7. The number of grids on the fluid surface (exhaust vent) should not be less than 4-5.

8. Select the corresponding grid spacing according to the actual size of the calculation object. On the basis of satisfying the grid is sufficiently fine, reduce the number of grids as much as possible to reduce the amount of calculation and improve the stability of convergence. According to the meshing instructions of Airpak2.1 software, about 1/40 of the size of the simulated laboratory is used as the basis for meshing, and on this basis, other physical or dense or sparse tissue grids in the room are used.

### 3.4.2. Control equation processing method

The continuity equation, momentum equation, energy equation, and indoor zero equation model for dealing with turbulence can be expressed in the following general form:

$$\rho div(\mu\phi) = div\ (\Gamma grad\phi) + S \tag{3-11}$$

Where:

$\phi$——Universal dependent variable, representing $(\mu, \varepsilon, \nu, \omega, \mathrm{T}, \kappa)$

ρ-density;

u ——speed vector;

Γ-generalized diffusion coefficient;

S-generalized source term

Based on formula (3-11) the control volume method is used for dispersion, and the steady-state transmission conservation equation of scalar ϕ is considered. At any control volume V, the integral control equation is as follows:

$$\oint \rho\phi\nu \cdot dA = \oint \Gamma_\phi \nabla\phi \cdot dA + \oint_v S_\phi dV \tag{3-12}$$

On a given grid, equation (3-12) is discrete as:

$$\sum_{f}^{N_{faces}} \nu_f \phi_f \mathrm{A}_f = \sum_{f}^{N_{faces}} \Gamma_\phi (\nabla\phi)_n \mathrm{A}_f + S_\phi V \tag{3-13}$$

The discrete equation can be written in the following general form:

$$a_p\phi = \sum_{nb} a_{nb}\phi_{nb} + b \tag{3-14}$$

The subscript *nb* in the equation represents the adjacent grid, $a_p$ and $a_{nb}$ are the coefficients of ϕ and nbϕ, respectively, and *b* is the source term.

### 3.4.3. Calculation method

In this chapter, the SIMPLE (Semi-Implicit Method for Pressure-Linked Equations) algorithm calculation steps are as follows (shown in Figure 3-5):

1. Estimate the pressure field $p^*$
2. Solve the system of momentum equations to get $\mu^*, \nu^*, \omega^*$
3. Solve the $p$ 'equation.
4. Calculate $p$ by adding $p*$ to $p$ '.

5. Use the speed correction formula to calculate the speed with an asterisk, $\mu$ , $\nu$ , $\omega$
6. Solve the discretization equations of other physical quantities $\phi$ that affect the flow field through source terms, fluid properties, etc.
7. Process the corrected pressure $p$ into a new estimated pressure $p$ *, return to the second step, repeat the whole process until the end of the whole process, and find the convergent solution.

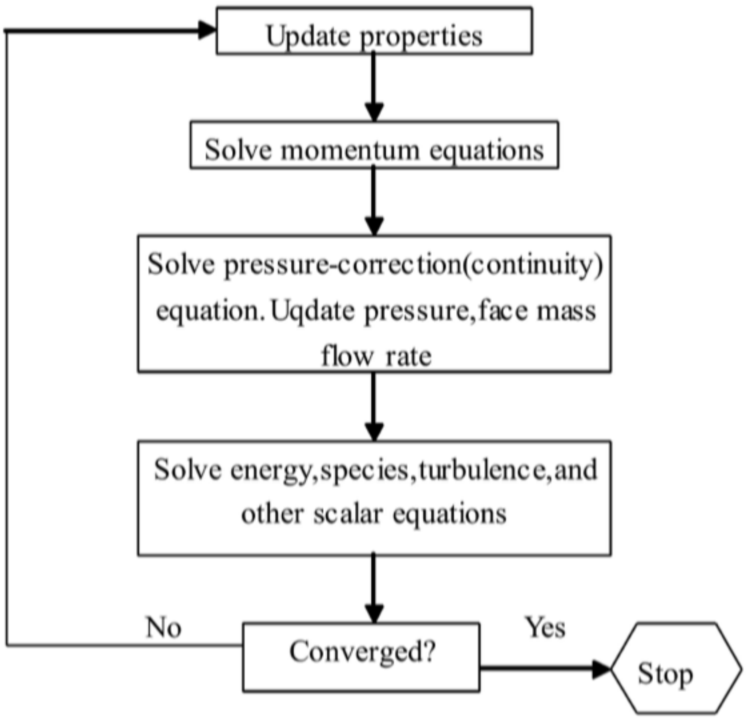



Figure 3-5 Solution process diagram

### 3.4.4. Distribution hypothesis and convergence principle of dependent variable

1. Distribution hypothesis of dependent variable
This topic adopts different distribution hypotheses for different dependent variables. When the grid is sufficiently fine, the variation of the dependent variable between the grids decreases, and the impact of such different assumptions will also be reduced. According to different calculation actual situations, the first-order or second-order welcome style is adopted to ensure the accuracy and stability of the calculation results.
2. The principle of convergence
The convergence criterion for the energy equation is generally taken as: $1 \times 10^{-6}$ (for example: the convergence criterion for temperature); the convergence criterion for the flow equation is generally taken as: $1 \times 10^{-3}$ (for example: the convergence criterion for velocity). The convergence criterion also reflects the accuracy of the simulation calculation.
3. Determination of relaxation factor

In the process of calculating by successive iteration method, the choice of relaxation factor is a more critical link. A properly selected relaxation factor can accelerate convergence. Improper selection may cause calculation oscillation or divergence. As far as the solution of the entire nonlinear problem is concerned, there is no complete theory to judge whether the iterative solution method can obtain a convergent solution. The sub-relaxation iteration method is used in FLUENT software to accelerate convergence. Many scholars have studied the determination of the optimal relaxation factor, but they all aim to directly find the optimal relaxation factor. The selection of such relaxation factors is quite complicated. The common method in engineering is to choose different relaxation factors for trial calculation according to the convergence speed of the iteration process, constantly modify, and gradually find the best, until it is satisfied, then fix it and continue the iteration, in order to achieve the purpose of accelerating convergence.

In the calculations in this chapter, the pressure relaxation factor is between 0.1 and 0.3, the momentum relaxation factor is between 0.1 and 0.5, the temperature and viscosity relaxation factor is between 0.7 and 1.0, and the mass force relaxation factor is 0.1. The relaxation factor of the turbulent dissipation rate is 0.3 to 0.5. Based on the microscopic analysis of the characteristics of air turbulence, the main method is to adopt the indoor zero equation model and use the controlled volume dispersion method to discrete the control equations. The professional software Airpak2.1 introduced by Fluent is used for simulation calculation. Airpak2.1 uses FLUENT5.6.6 CFD solver for calculation.

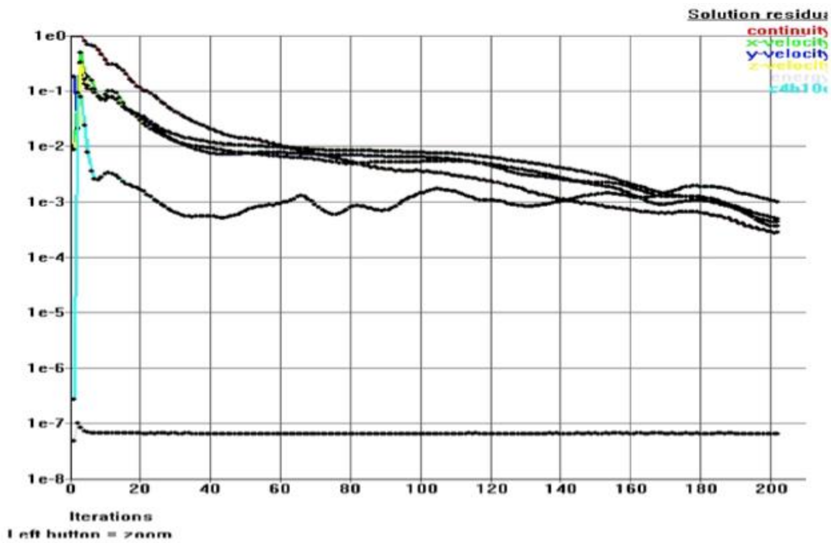



Figure 3-6 Convergence process diagram

The unstructured grid is used to divide the three-dimensional model, and the regions with large temperature and humidity gradient changes at the inlet and the fan outlet are grid-encrypted. To ensure that the number of meshes used in the simulation is not related to the corresponding simulation results, a mesh independence verification was performed. Models with three grids A, B, and C, with numbers of 2500984, 2700880, and 300980, were selected for simulation. In case 1, select a total of ten simulation points for the air inlet, air outlet, humidifier outlet and indoor as reference. The average humidity values at various measurement points were compared and three models of A, B, and C were less than 5 %. As shown in Figure 3-7. It shows that the grid encryption based on model A has little effect on the simulation results. Considering the calculation rate, A model was selected for simulation analysis. The maximum plane twist rate of the grid is 0.6, and 97.3% of the grid twist rates are less than 0.5. The grid quality is good and meets the simulation requirement.

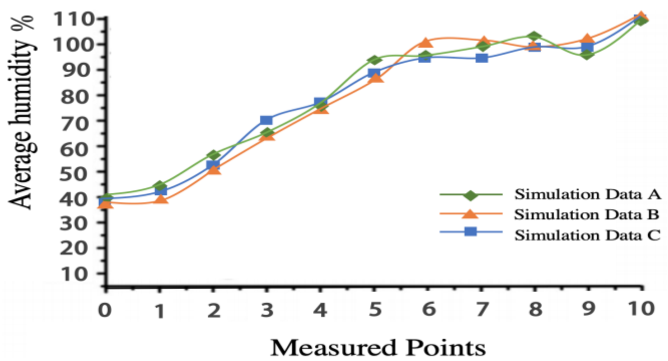



Figure 3-7 Independence of result within grid meshes

## 3.4.5. Velocity field analysis (air flow analysis)

### 3.4.5.1. Velocity field analysis (air flow analysis) of case 1

The object of this simulation is mainly the indoor humidity distribution, so the x, y-axis cross-section and the ground 0.2m height cross section and the human standing height 1.5m cross section at the humidifier are analyzed。

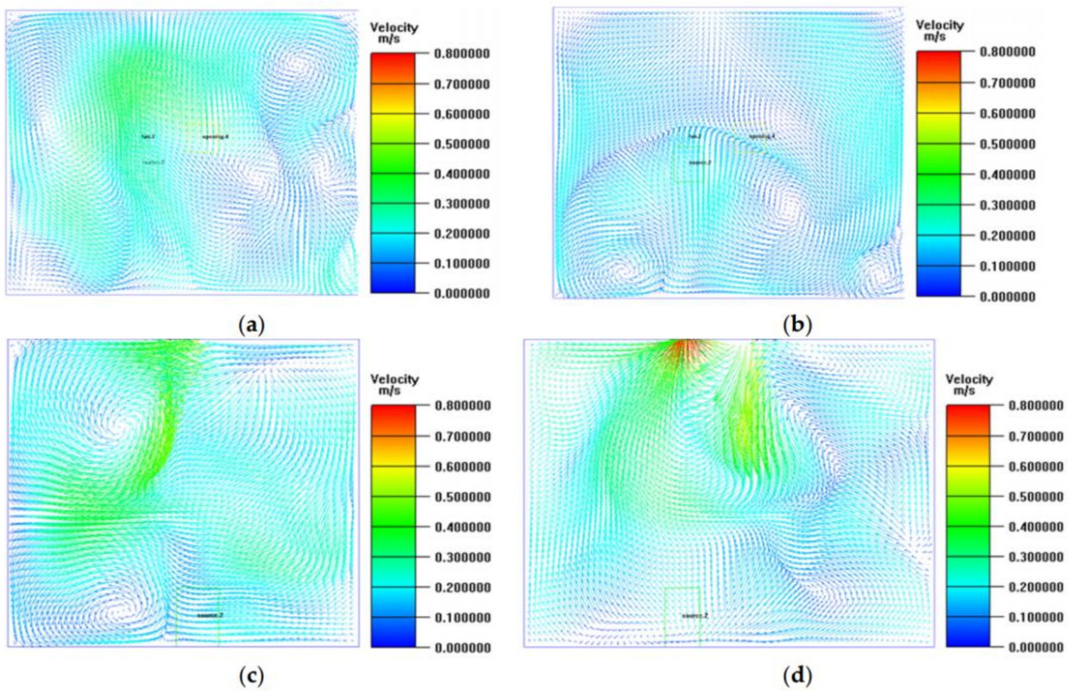



Figure 3-8 Velocity field of axial section of Case 1: (a) air velocity field of horizontal cross-section of Case 1 (Z = 1.5 m); (b) air velocity field of horizontal cross-section of Case 1 (Z = 0.2 m); (c) x-axial center cross-section velocity field; (d) y-axial ross-section velocity field

Figure 3-8 are the velocity field diagram of the near-ground height of 0.2m and the human standing working height of 1.5m of CASE1.Case1 is that the air return is in the ceiling. It can be seen from the figure that the airflow enters the model from the ceiling, which affects the indoor humidity distribution and forms a vortex around the humidifier.

Figure 3-8 The velocity field diagram of the central cross-section of the X-axis. It can be seen from the figure that part of the water vapor emitted from the generating source rises due to its initial velocity and forms a vortex with the airflow entering the air supply port, and the other part is discharged outside the room through the exhaust port.

Figure 3-8 shows The velocity field diagram of the central cross-section of theY-axis. It can be seen from the figure that part of the water vapor emitted from the generating source rises due to its initial velocity and forms a vortex with the airflow entering the air supply port, and the other part is discharged outside the room through the exhaust port.

After the dilution effect, it is discharged outside the room, and the other part is directly taken out of the laboratory by the exhaust port due to the jet induction effect of the exhaust port. The naturally ventilated window and the exhaust port are located on the same side of the laboratory. Due to this jet induced effect of the exhaust port, the "window-exhaust port" forms a backflow, and the amount of fresh air entering from the window is mostly free of wet air It is diluted and discharged directly out of the room.

Based on the above analysis, it can be concluded that the general distribution of the entire indoor fluid flow field is basically a part of the air sent from the air supply port to form two large vortices between the humidifier and the end of the exhaust port; Due to the jet induction of the exhaust port, it is directly discharged outside. The general jet formula is difficult to give the position of the vortex, and the CFD software can clearly show the specific characteristics such as position and shape. The jet formula often used in engineering design cannot consider the influence of the boundary, nor

The influence between jets can be considered, and CFD results can reflect these influences, which is also the advantage of CFD in the analysis of airflow organization.

### 3.4.5.2. Velocity field analysis (air flow analysis) of case 2

The object of this simulation is mainly the indoor humidity distribution, so the x, y-axis cross-section and the ground 0.2m height cross section and the human standing height 1.5m cross section at the humidifier are analyzed。

Figure 3-9 are the velocity field diagram of the near-ground height of 0.2m and the human standing working height of 1.5m of CASE 2.

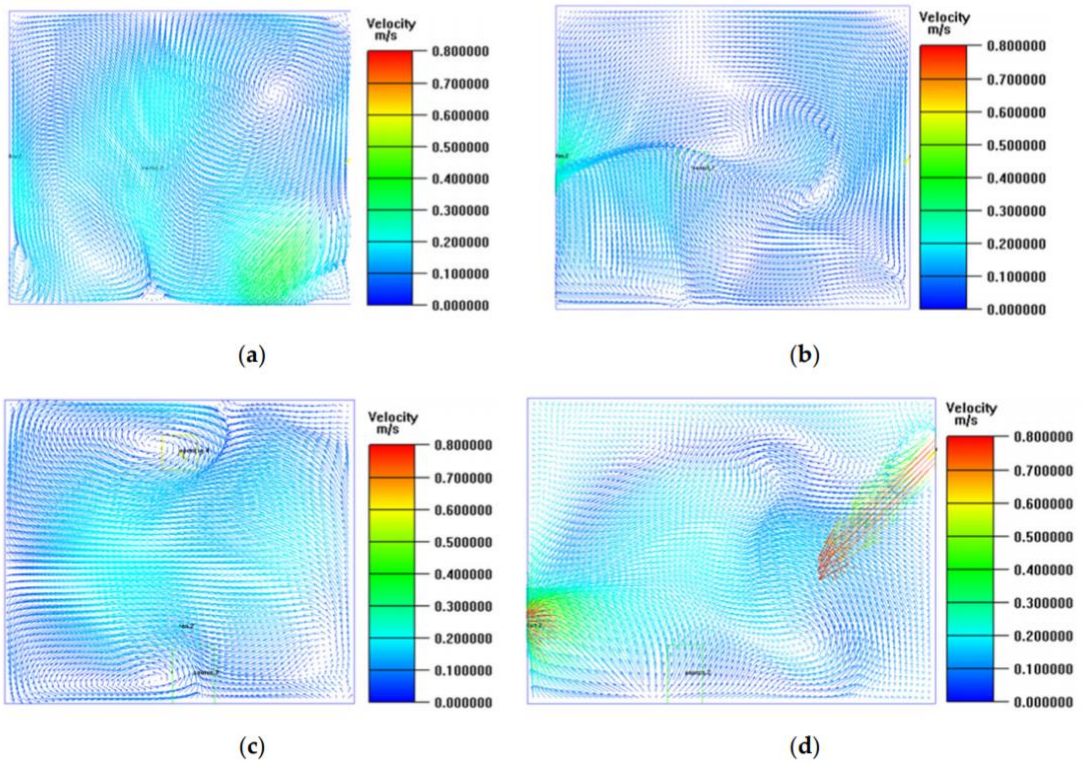



Figure 3-9 Velocity field of axial section of Case 2: (a) air velocity field of horizontal cross-section of Case 2 (Z = 1.5 m); (b) air velocity field of horizontal cross-section of Case 2 (Z = 0.2 m); (c) x-axial center cross-section velocity field; (d) y-axial ross-section velocity field

Figure 3-9 also shows The velocity field diagram of the central cross-section of the x-axis. It can be seen from the figure that part of the water vapor emitted from the generating source rises due to its initial velocity and forms a vortex with the airflow entering the air supply port, and the other part is discharged outside the room through the exhaust port.

After the dilution effect, it is discharged outside the room, and the other part is directly taken out of the laboratory by the exhaust port due to the jet induction effect of the exhaust port. The naturally ventilated window and the exhaust port are located on the same side of the laboratory. Due to this jet induced effect of the exhaust port, the "window-exhaust port" forms a backflow, and the amount of fresh air entering from the window is mostly free of wet air It is diluted and discharged directly out of the room.

Based on the above analysis, it can be concluded that the general distribution of the entire indoor fluid flow field is basically a part of the air sent from the air supply port to form two large vortices between the humidifier and the end of the exhaust port; Due to the jet induction of the exhaust port, it is directly discharged outside. The general jet formula is difficult to give the position of the vortex, and the CFD software can clearly show the specific characteristics such as position and shape. The jet formula often used in engineering design cannot consider the influence of the boundary, nor

The influence between jets can be considered, and CFD results can reflect these influences, which is also the advantage of CFD in the analysis of airflow organization.

### 3.4.5.3. Velocity field analysis (air flow analysis) of case 3

The object of this simulation is mainly the indoor humidity distribution, so the x, y-axis cross-section and the ground 0.2m height cross section and the human standing height 1.5m cross section at the humidifier are analyzed。

Figure 3-10 are the velocity field diagram of the near-ground height of 0.2m and the human standing working height of 1.5m of CASE 3.

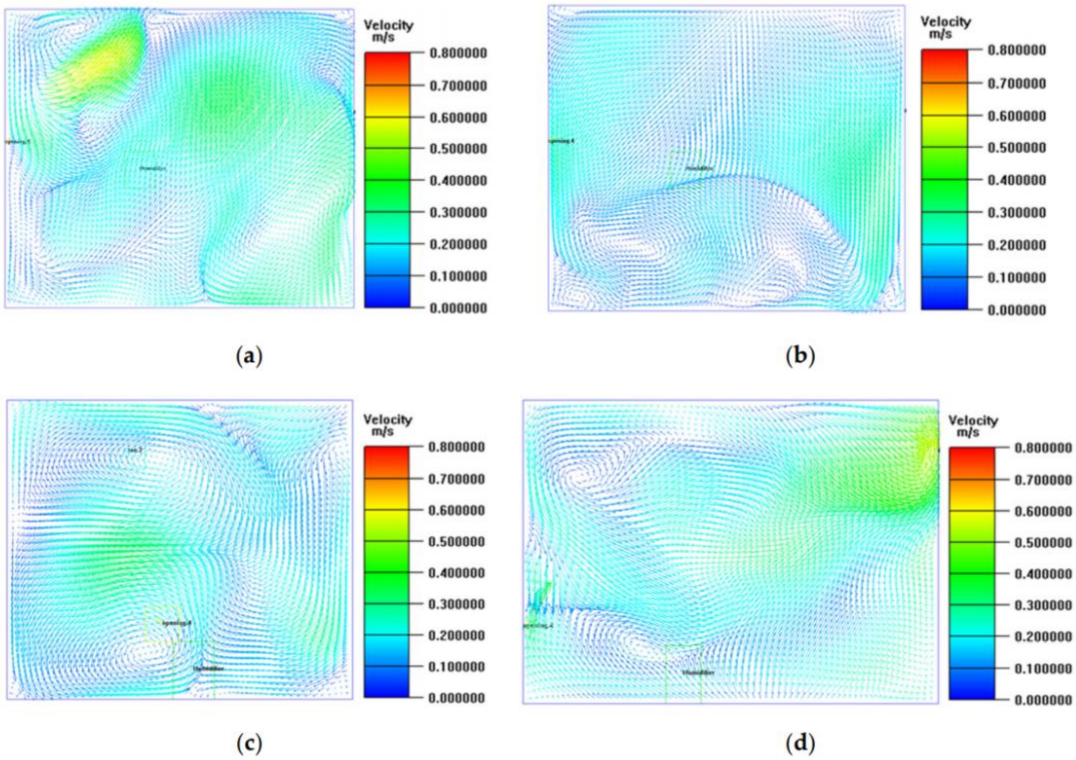



Figure 3-10 Velocity field of axial section of Case 3: (a) air velocity field of horizontal cross-section of Case 3 (Z = 1.5 m); (b) air velocity field of horizontal cross-section of Case 3 (Z = 0.2 m); (c) x-axial center cross-section velocity field; (d) y-axial cross-section velocity field

Figure 3-10 The velocity field diagram of the central cross-section of the X-axis. It can be seen from the figure that part of the water vapor emitted from the generating source rises due to its initial velocity and forms a vortex with the airflow entering the air supply port, and the other part is discharged outside the room through the exhaust port.

Figure 3-10 shows The velocity field diagram of the central cross-section of the Y-axis. It can be seen from the figure that part of the water vapor emitted from the generating source rises due to its initial velocity and forms a vortex with the airflow entering the air supply port, and the other part is discharged outside the room through the exhaust port.

After the dilution effect, it is discharged outside the room, and the other part is directly taken out of the laboratory by the exhaust port due to the jet induction effect of the exhaust port. The naturally ventilated window and the exhaust port are located on the same side of the laboratory. Due to this jet induced effect of the exhaust port, the "window-exhaust port" forms a backflow, and the amount of fresh air entering from the window is mostly free of wet air It is diluted and discharged directly out of the room.

Based on the above analysis, it can be concluded that the general distribution of the entire indoor fluid flow field is basically a part of the air sent from the air supply port to form two large vortices between the humidifier and the end of the exhaust port; Due to the jet induction of the exhaust port, it is directly discharged outside. The general jet formula is difficult to give the position of the vortex, and the CFD software can clearly show the specific characteristics such as position and shape. The jet formula often used in engineering design cannot consider the influence of the boundary. The influence between jets can be considered, and CFD results can reflect these influences, which is also the advantage of CFD in the analysis of airflow organization.

### 3.4.5.4. Velocity field analysis (air flow analysis) of case 4

The object of this simulation is mainly the indoor humidity distribution, so the x, y-axis cross-section and the ground 0.2m height cross section and the human standing height 1.5m cross section at the humidifier are analyzed。

Figure 3-11 are the velocity field diagram of the near-ground height of 0.2m and the human standing working height of 1.5m of CASE 4.

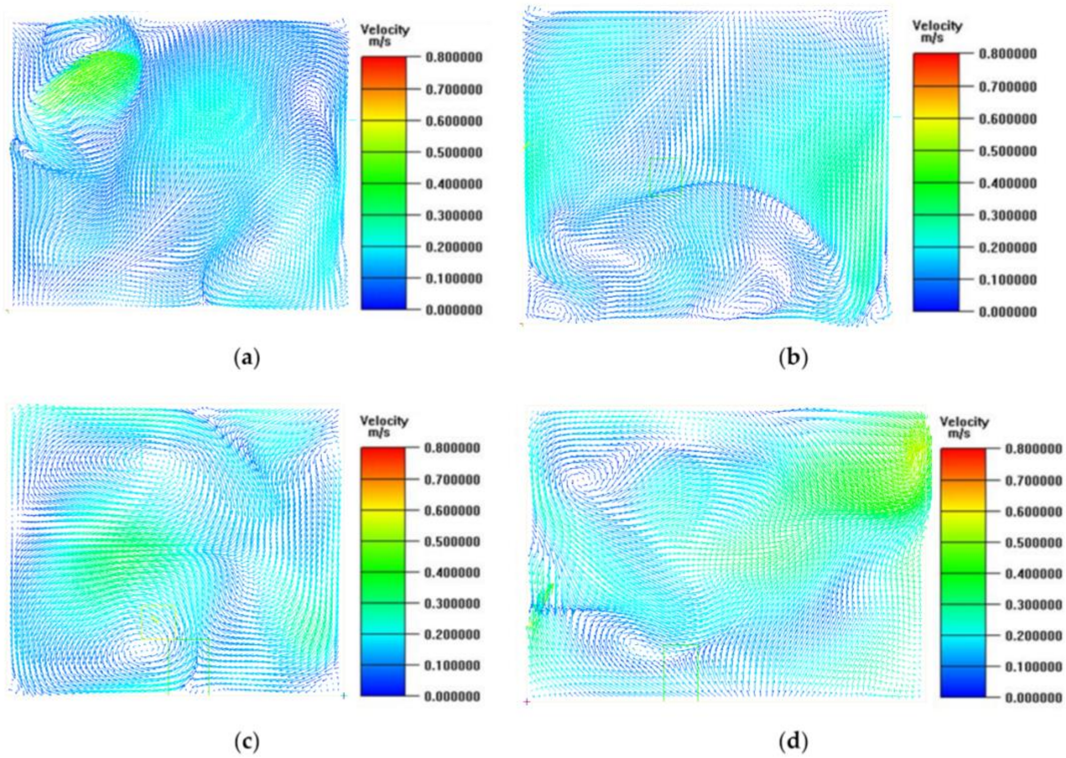



Figure 3-11 Velocity field of axial section of Case 4: (a) air velocity field of horizontal cross-section of Case 4 (Z = 1.5 m); (b) air velocity field of horizontal cross-section of Case 4 (Z = 0.2 m); (c) x-axial center cross-section velocity field; (d) y-axial cross-section velocity field

As only the wall material of the room and the amount of humidifier humidification were changed between case3 and case4, there was no significant difference in the velocity distribution in the room.

After the dilution effect, it is discharged outside the room, and the other part is directly taken out of the laboratory by the exhaust port due to the jet induction effect of the exhaust port. The naturally ventilated window and the exhaust port are located on the same side of the laboratory. Due to this jet induced effect of the exhaust port, the "window-exhaust port" forms a backflow, and the amount of fresh air entering from the window is mostly free of wet air It is diluted and discharged directly out of the room.

Based on the above analysis, it can be concluded that the general distribution of the entire indoor fluid flow field is basically a part of the air sent from the air supply port to form two large vortices between the humidifier and the end of the exhaust port; Due to the jet induction of the exhaust port, it is directly discharged outside. The general jet formula is difficult to give the position of the vortex, and the CFD software can clearly show the specific characteristics such as position and shape. The jet formula often used in engineering design cannot consider the influence of the boundary. The influence between jets can be considered, and CFD results can reflect these influences, which is also the advantage of CFD in the analysis of airflow organization.

### 3.4.6. Relative humidity analysis

Figure 3-11 shows the relative humidity distribution of the central section under the same ventilation and different ventilation modes. The range of relative humidity RH ≥ 100% in the figure characterizes the formation area of the water mist. It can be seen from Figure 2 that the relative humidity distribution of the central section is related to the air supply mode, and the condensation state by different air supply modes is different; Compared with other cases, the relative humidity value of Case 1 is generally low; the high humidity area in Case 2 space is distributed the lower space; the high humidity area in Case 3 space is concentrated the upper space. Because of the wall is insulation, the Case 4 almost no condensation near the wall space. And the condensation space is concentrated the upper space of humidifier. Analysis of the water mist formed near the wet source outlet. The results show that the water mist formed in Case 1 is concentrated in the range of 0.5- 0.7m above the wet source outlet; the water mist formed in Case 2 is concentrated in the upper right of the wet source outlet in the range of 0.5-1.1 m, and the water mist is the largest. Case 3 forms the smallest volume of water mist, which is 0.5m above the wet source outlet. Case 3 forms the smallest volume of water mist, which is 0.2 to 0.3 m above the wet source outlet. Comparing Case 4 with Case 3, since Case 4 increases the humidifier's humidification amount, the condensation area in the area above the humidifier is larger than that of Case 3. The humidity distribution of Case 2 and Case 3 is obvious differences. The relative humidity distribution of the central section under the same ventilation and different ventilation modes. The range of relative humidity at RH ≥ 100% in the figure characterizes the formation of water mist. It can be seen from Figure 6 that the relative humidity distribution of the central section is related to the air supply mode, and the condensation state by different air supply modes is different. Comparing the cases, the relative humidity value of Case 1 was generally low, while Case 2 had a high humidity area in the lower space, and Case 3 had a high humidity area in the upper space. Due to the wall insulation included in Case 4, almost no condensation appeared near the wall space, and the condensation space was concentrated in the upper space of the humidifier

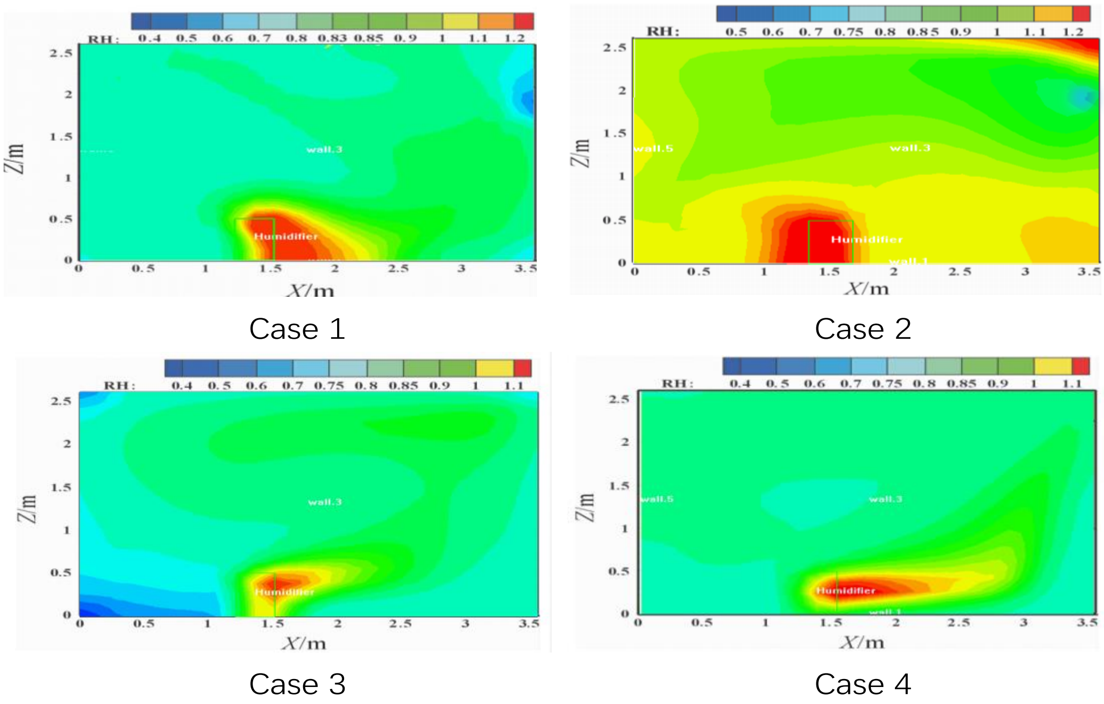



Figure 3-12 The Sectional Humidity Distribution

The results of the analysis of the water mist formed near the wet source outlet showed that the water vapor formed in Case 1 was concentrated at 0.5–0.7 m above the wet source outlet; the water mist formed in Case 2 was concentrated in the upper right of the wet source outlet at 0.5–1.1 m, and the quantity of water mist was greater for Case 2. Case 3 formed the smallest volume of water mist, which was concentrated 0.5 m above the wet source outlet. Comparing Case 4 with Case 3, since Case 4 increased the amount of humidification in the humidifier, the condensation area in the area above the humidifier was larger than in Case 3. There were obvious differences in humidity distribution between Case 2 and Case 3. Figure 3-12 shows the temperature distribution of Case 2 and Case 3. It can be seen that when the air is supplied from the lower part of the room, the hot air filled the whole area

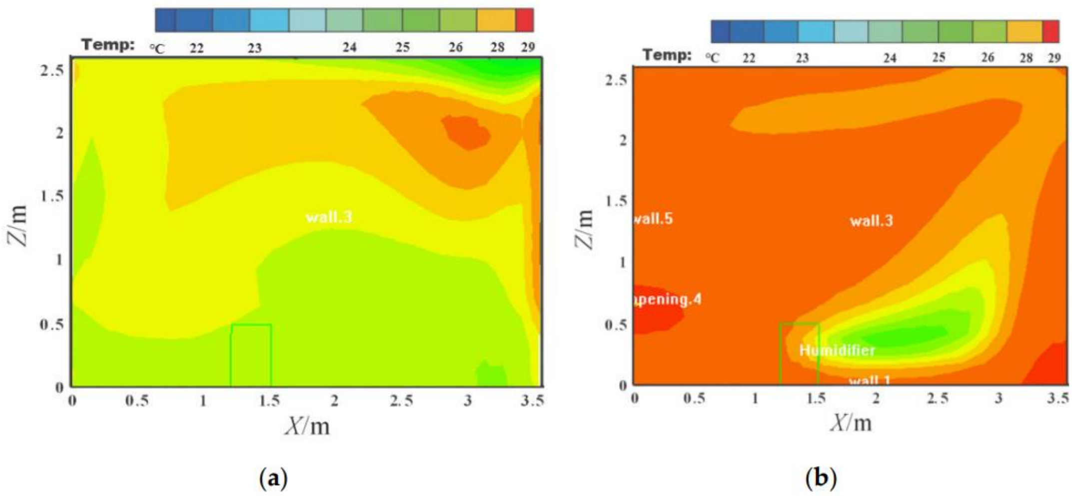



Figure 3-13 Sectional temperature distribution in (a) Case 2 and (b) Case 3.

To study the effects of ventilation on space and wall condensation, the intersection of the wet source exit position from the ceiling (recorded as P1 line), and the wall X=0 and the wall Y=3m (referred to as P2 line) are the research objects. The variation of the condensation rate of space and wall along the height direction under different ventilation modes was analyzed. The position of P1 line and P2 line is shown in Figure 1(Case 1). From Figure 3-13 in the case of the same ventilation, the ventilation method of Case 3 has the best effect on the condensation of the space above the wet source outlet, except for a little condensation; the ventilation method of Case 2 is the most unfavorable for control, and the condensation in the upper space occurs. The highest rate means that the time of formation of water mist in the space above the wet source is short, the speed is fast, and the amount of condensed water is large. In the case of the same ventilation, the ventilation method of case 3 has the best effect on the condensation of the space above the wet source outlet, except for a little condensation in the range of 0.4-0.6 m in height, and condensation at the other positions. The rate is 0, no condensation occurs; the ventilation method of case 2 is the most unfavorable for control, and the condensation in the upper space occurs. From the condensation rate distribution curve, the condensation of case 2 is in the range of 0.7 to 2.1 m in height. The highest rate means that the time of formation of water mist in the space above the wet source is short, the speed is fast, and the amount of condensed water is large.

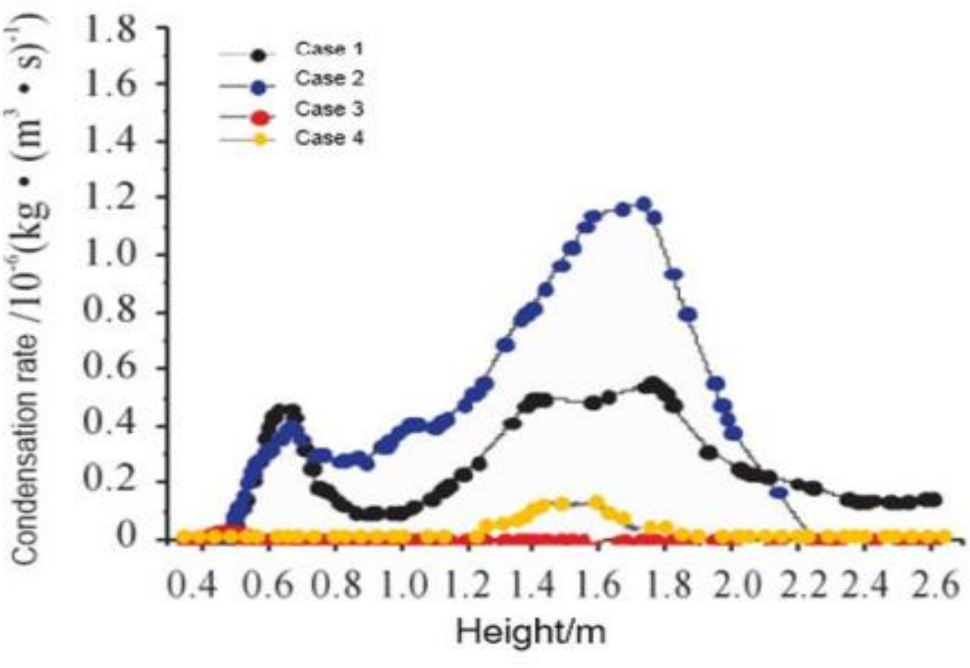



Figure 3-14 Condensation Rate Changes Along The P1 Line Under Different Air Supply Mode.

It can be seen from the Figure 3-14 that in Case1, the wall condensation rate increases with the increase of the wall height. This means that the condensation at the top position is early, fast, and has a large amount of condensation. And condensation formation at the bottom of the room is late, slow, and less condensation. The Case 2 and Case 3 condensation rate curves are similar, and the condensation rate of the wall decreases with the increase of the height. At this time, the closer to the wall foot position, the faster the condensation is formed and the more the condensation is. Comparing the three working conditions, it can be seen that the space and wall condensation rate of case 3 is the lowest when the same ventilation amount is used. Therefore, the ventilation of Case 3 is optimal from the viewpoint of reducing the condensation rate of the wall。And that in case1, the wall condensation rate increases with the increase of the wall height, which means that the condensation formation time is earlier, the speed is faster, and the condensation is larger; the condensation formation time below the wall surface is late and the speed Slow, less condensation. The case2 and case3 condensation rate curves are similar, and the condensation rate of the wall decreases with the increase of the height. At this time, the closer to the wall foot position, the faster the condensation is formed and the more the condensation is. Comparing the three working conditions, it can be seen that in the case of the same ventilation, the condensation rate of the space and the wall surface of Case 3 is the lowest. Therefore, the ventilation mode of Case 3 is optimal from the viewpoint of reducing the condensation rate of the wall surface.

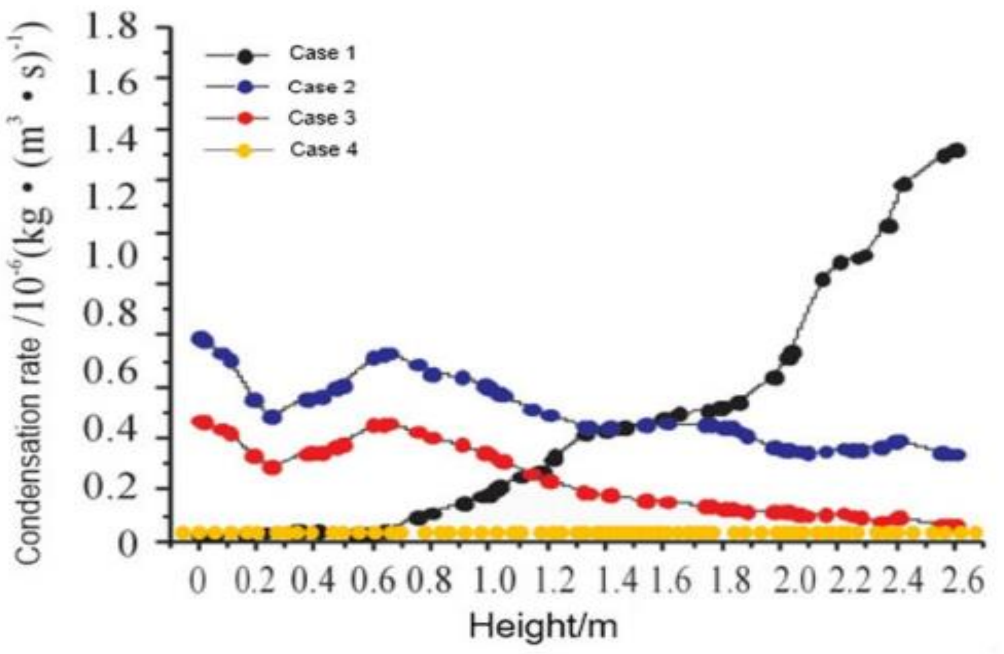


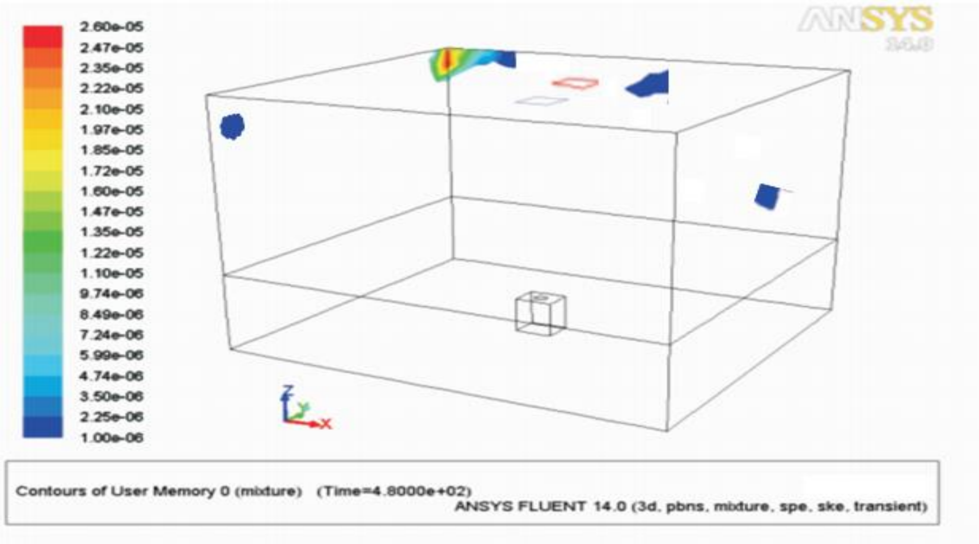

Figure 3-15 Condensation Rate Changes Along The P2 Line Under Different Air Supply Mode.

## 3.5. Simulation results of the influence of ventilation on wall condensation and analyses

Figures 2-17 is simulations of wall condensation distribution changes at typical times under different operating conditions. The values of the left strip color bands in the figure characterize the condensation of the wall in kg. It can be seen from Figure 2 that when the working case 1, the time at which the wall surface starts to condensation is 480s, and the initial occurrence position of the condensation is the corner of the ceiling near the side of the air outlet, and the wall surface is stable from the occurrence of condensation to the condensation area. With a duration of 1800s, the maximum condensation on the wall is 0.33g; at the case 2, the initial time of condensation is 600s, and the initial occurrence of condensation is the ceiling corner and the two side walls near the air supply side. In the middle area, the wall surface lasts for 1800s from the time when condensation occurs and the condensation area is stable, and the maximum condensation on the wall is 1.14g. At case 3, the initial time of condensation is 900s, the occurrence position is the corner of the floor near the side of the air supply opening. The time from the occurrence of dew condensation to the stable condensation area is 2400 s, and the maximum condensation on the wall is 0.299 g.

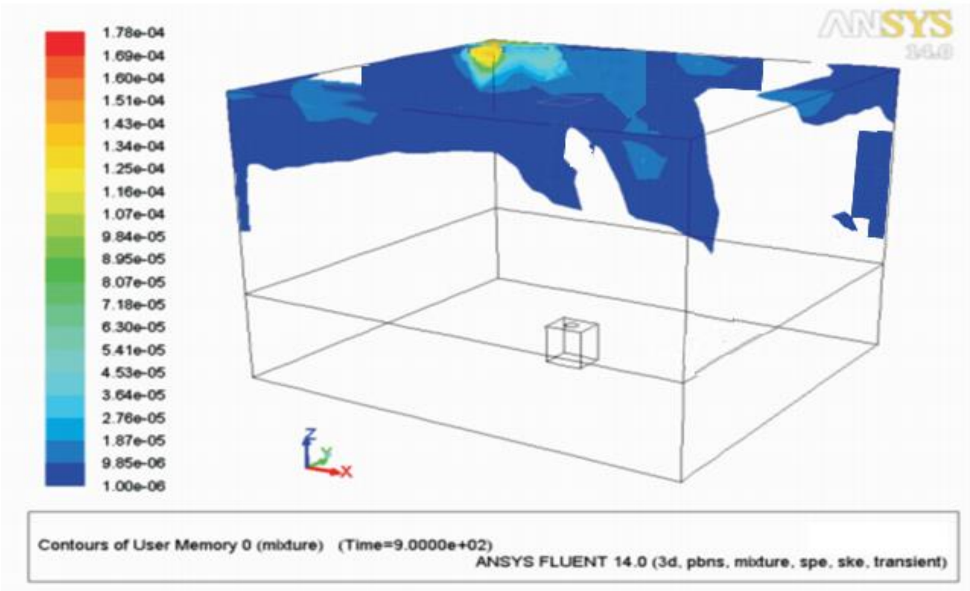
From the distribution of the condensation on the wall, in the ventilation mode of the roof to the roof, the condensation starts at the corner of the ceiling near the side of the exhaust, and then appears at the intersection of the ceiling and the surrounding walls and the corners. Condensation phenomenon, with the continuous generation of hot steam, the condensation area began to gradually extend from the ceiling to the bottom of the wall until the condensation area stabilized.

In the ventilation mode of the row, the dew condensation starts at the side of the ceiling wall near the air supply opening and the middle area of the left and right wall surfaces, and then the condensation occurs on the surrounding wall corner of the floor and the side wall surface of the air supply opening, and then the condensation area begins. Extend until it stabilizes. In the ventilation mode of the upper row on the lower side of the delivery side, the dew condensation starts at the sides of the bottom corner of the air supply opening, and then condensation occurs on the left and right side wall surfaces of the air supply opening and the corner of the ceiling wall on the side of the air outlet. The area of condensation began to expand until it stabilized. Comparing the condensation conditions under various ventilation modes, it can be known that when the ventilation amount is the same but the ventilation mode is different, the condensation of each wall surface is very different. It can be seen from the figure that the condensation area of case 3 is the smallest, and the working condition is the second, and the working case 2 has the largest dew area. Therefore, from the perspective of reducing the risk of wall condensation, the ventilation mode of case 3 is optimal

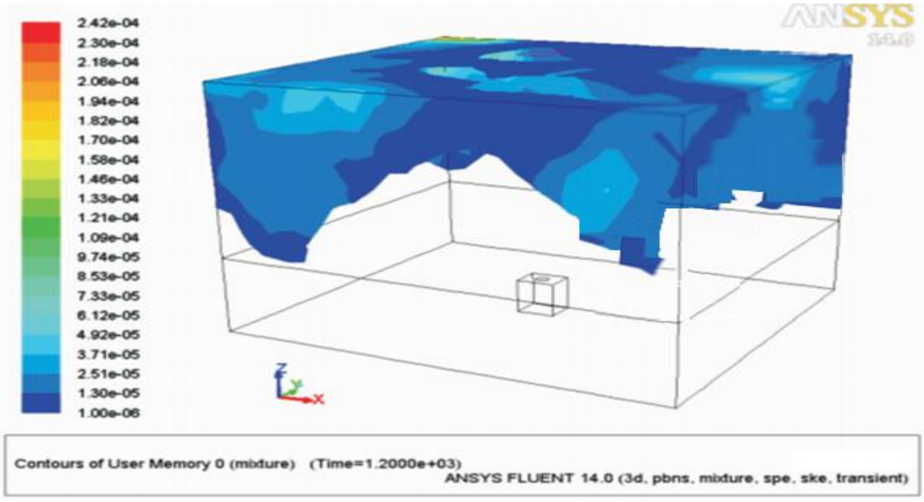

2.60e-05
2.47e-05
2.35e-05
2.22e-05
2.10e-05
1.97e-05
1.85e-05
1.72e-05
1.60e-05
1.47e-05
1.35e-05
1.22e-05
1.10e-05
9.74e-06
8.49e-06
7.24e-06
5.99e-06
4.74e-06
3.50e-06
2.25e-06
1.00e-06
Contours of User Memory 0 (mixture) (Time=4.8000e+02)
ANSYS FLUENT 14.0 (3d, pbns, mixture, spe, ske, transient)

.

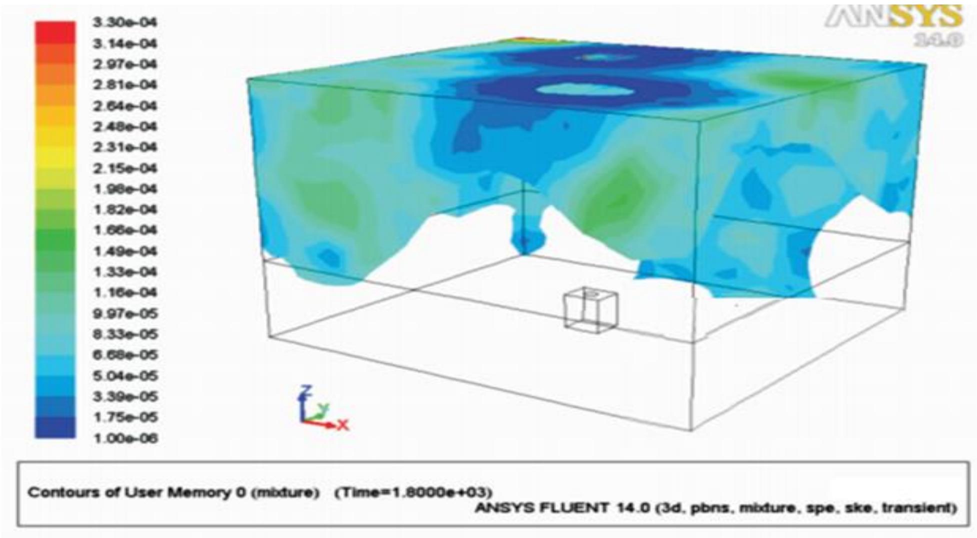
(a)


1.78e-04
1.69e-04
1.60e-04
1.51e-04
1.43e-04
1.34e-04
1.25e-04
1.16e-04
1.07e-04
9.84e-05
8.95e-05
8.07e-05
7.18e-05
6.30e-05
5.41e-05
4.53e-05
3.64e-05
2.76e-05
1.87e-05
9.85e-06
1.00e-06
Contours of User Memory 0 (mixture) (Time=9.0000e+02)
ANSYS FLUENT 14.0 (3d, pbns, mixture, spe, ske, transient)

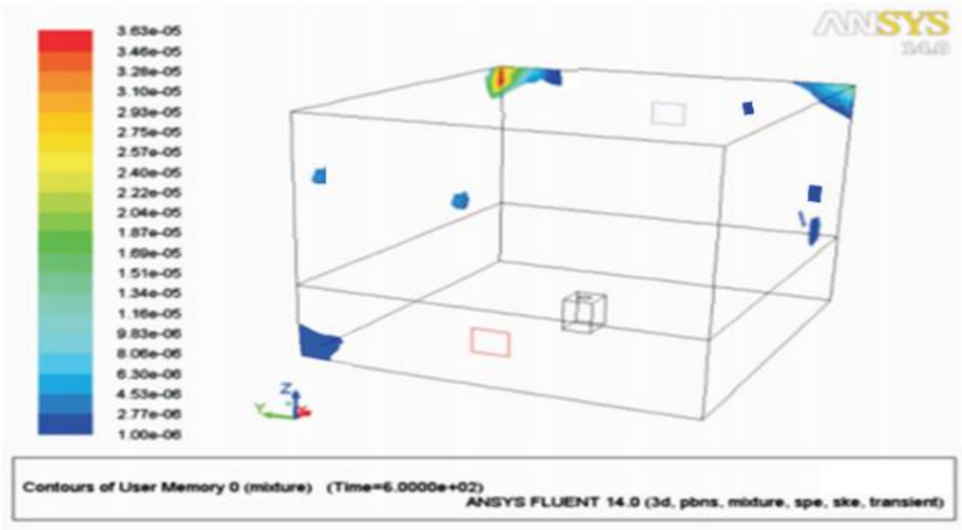
( b)


2.42e-04
2.30e-04
2.18e-04
2.06e-04
1.94e-04
1.82e-04
1.70e-04
1.58e-04
1.46e-04
1.33e-04
1.21e-04
1.09e-04
9.74e-05
8.53e-05
7.33e-05
6.12e-05
4.92e-05
3.71e-05
2.51e-05
1.30e-05
1.00e-06
Contours of User Memory 0 (mixture) (Time=1.2000e+03)
ANSYS FLUENT 14.0 (3d, pbns, mixture, spe, ske, transient)

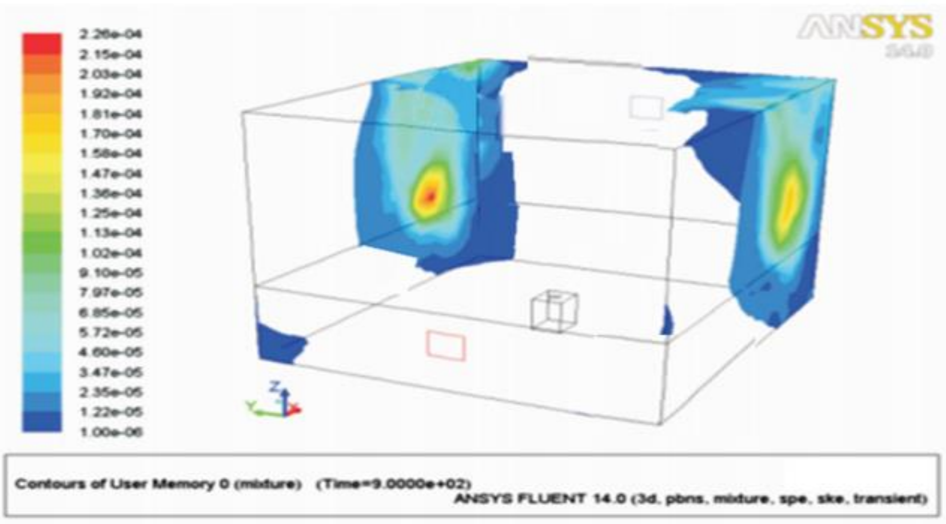
( c)


Contours of User Memory 0 (mixture) (Time=1.8000e+03)
ANSYS FLUENT 14.0 (3d, pbns, mixture, spe, ske, transient)

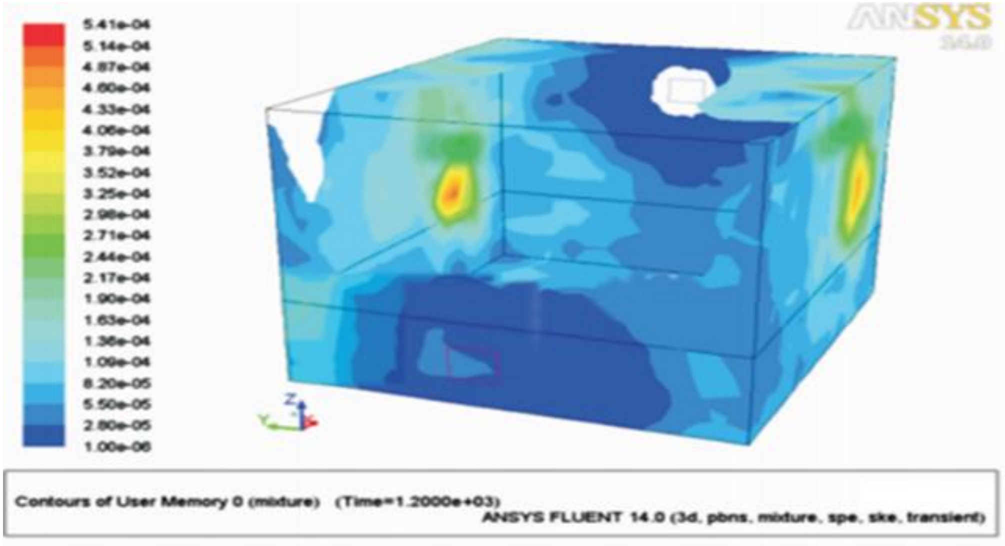
( d)


Contours of User Memory 0 (mixture) (Time=6.0000e+02)
ANSYS FLUENT 14.0 (3d, pbns, mixture, spe, ske, transient)

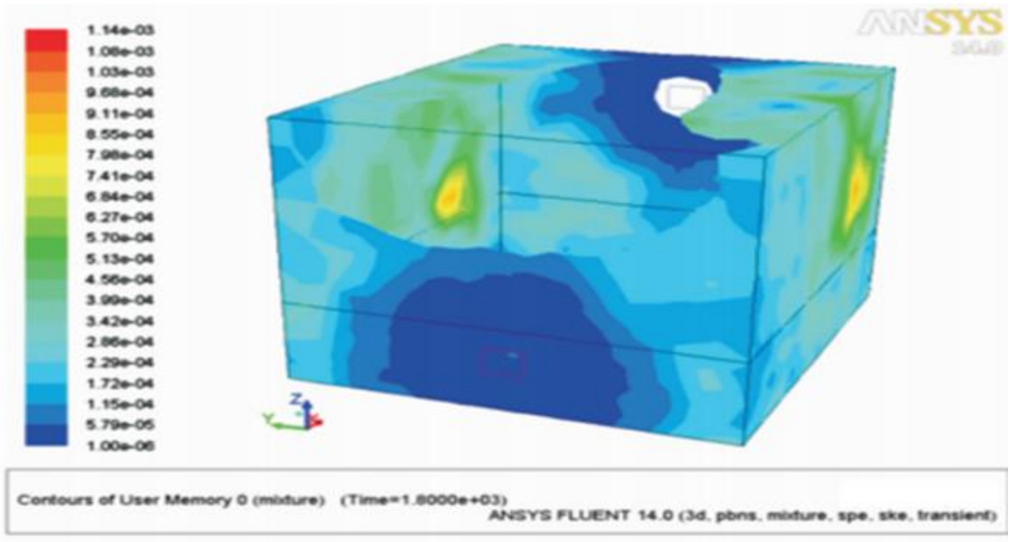
( e)


Contours of User Memory 0 (mixture) (Time=9.0000e+02)
ANSYS FLUENT 14.0 (3d, pbns, mixture, spe, ske, transient)

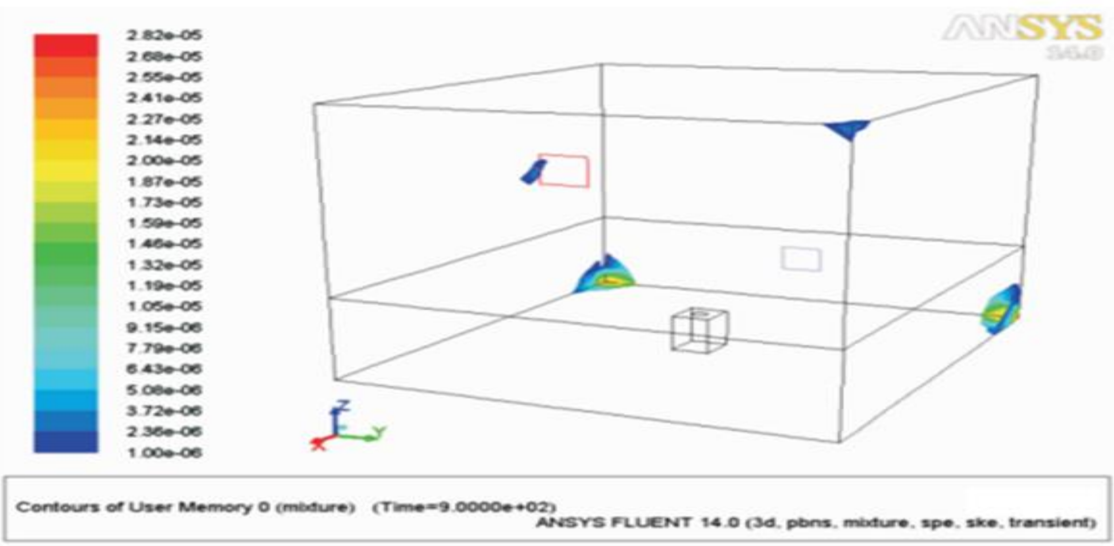
( f)


5.41e-04
5.14e-04
4.87e-04
4.60e-04
4.33e-04
4.06e-04
3.79e-04
3.52e-04
3.25e-04
2.98e-04
2.71e-04
2.44e-04
2.17e-04
1.90e-04
1.63e-04
1.36e-04
1.09e-04
8.20e-05
5.50e-05
2.80e-05
1.00e-06
Contours of User Memory 0 (mixture) (Time=1.2000e+03)
ANSYS FLUENT 14.0 (3d, pbns, mixture, spe, ske, transient)

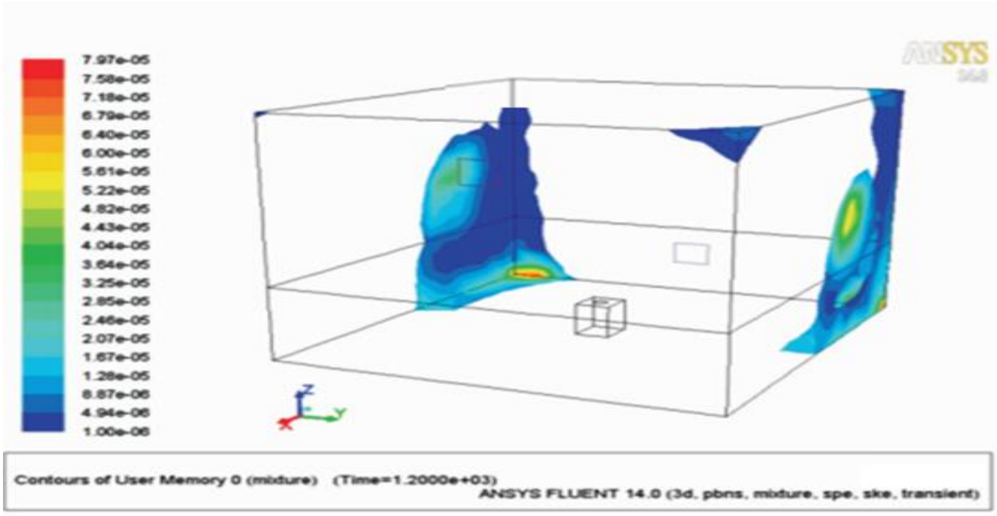
( g)


1.14e-03
1.08e-03
1.03e-03
9.68e-04
9.11e-04
8.55e-04
7.98e-04
7.41e-04
6.84e-04
6.27e-04
5.70e-04
5.13e-04
4.56e-04
3.99e-04
3.42e-04
2.85e-04
2.29e-04
1.72e-04
1.15e-04
5.79e-05
1.00e-06
Contours of User Memory 0 (mixture) (Time=1.8000e+03)
ANSYS FLUENT 14.0 (3d, pbns, mixture, spe, ske, transient)

( h)

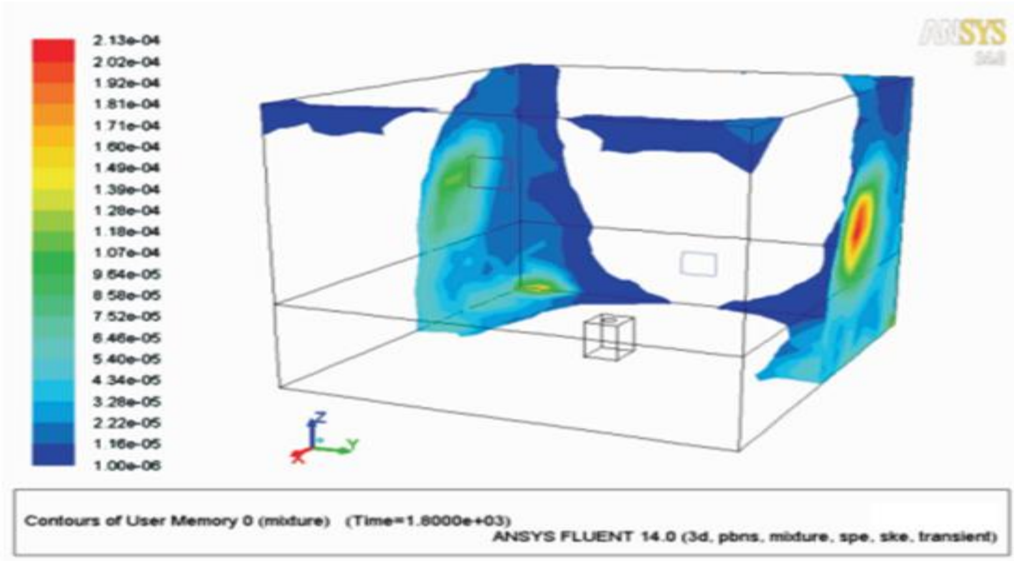

2.82e-05
2.68e-05
2.55e-05
2.41e-05
2.27e-05
2.14e-05
2.00e-05
1.87e-05
1.73e-05
1.59e-05
1.46e-05
1.32e-05
1.19e-05
1.05e-05
9.15e-06
7.79e-06
6.43e-06
5.08e-06
3.72e-06
2.36e-06
1.00e-06
Contours of User Memory 0 (mixture) (Time=9.0000e+02)
ANSYS FLUENT 14.0 (3d, pbns, mixture, spe, ske, transient)

(i)



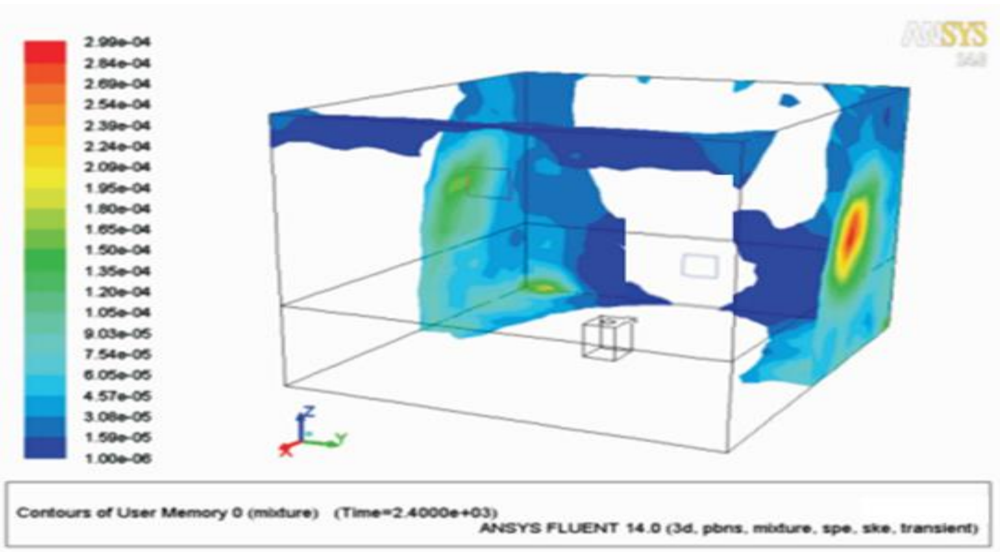
7.97e-05
7.58e-05
7.18e-05
6.79e-05
6.40e-05
6.00e-05
5.61e-05
5.22e-05
4.82e-05
4.43e-05
4.04e-05
3.64e-05
3.25e-05
2.85e-05
2.46e-05
2.07e-05
1.67e-05
1.28e-05
8.87e-06
4.94e-06
1.00e-06
ANSYS
Contours of User Memory 0 (mixture) (Time=1.2000e+03)
ANSYS FLUENT 14.0 (3d, pbns, mixture, spe, ske, transient)


( j)



2.13e-04
2.02e-04
1.92e-04
1.81e-04
1.71e-04
1.60e-04
1.49e-04
1.39e-04
1.28e-04
1.18e-04
1.07e-04
9.64e-05
8.58e-05
7.52e-05
6.46e-05
5.40e-05
4.34e-05
3.28e-05
2.22e-05
1.16e-05
1.00e-06
ANSYS
Contours of User Memory 0 (mixture) (Time=1.8000e+03)
ANSYS FLUENT 14.0 (3d, pbns, mixture, spe, ske, transient)


( k)



2.99e-04
2.84e-04
2.69e-04
2.54e-04
2.39e-04
2.24e-04
2.09e-04
1.95e-04
1.80e-04
1.65e-04
1.50e-04
1.35e-04
1.20e-04
1.05e-04
9.03e-05
7.54e-05
6.05e-05
4.57e-05
3.08e-05
1.59e-05
1.00e-06
ANSYS
Contours of User Memory 0 (mixture) (Time=2.4000e+03)
ANSYS FLUENT 14.0 (3d, pbns, mixture, spe, ske, transient)

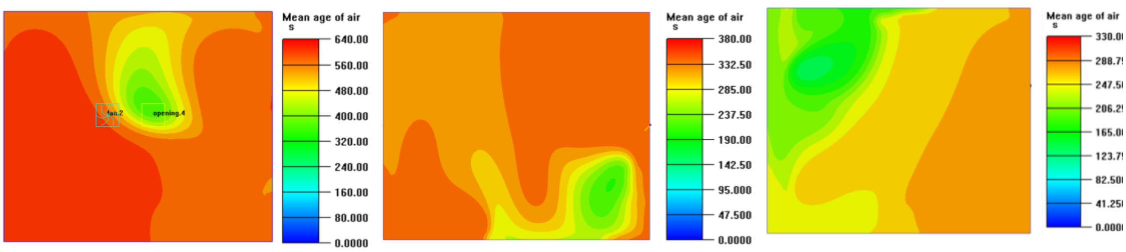

(l)

## 3.6. Air mean age analysis

Age of air is the age of air mass (Age of air), refers to the air mass since the entry into the room to a point in the room experienced time, reflecting the freshness of indoor air, it can be a comprehensive measure of the room ventilation effect, is an important indicator to evaluate the quality of indoor air. The concept of air age was first introduced by Sandberg in the 1980s. By definition, air age refers to the time when air enters a room. When pollutants are evenly distributed in a room and the air supply is fully fresh, the smaller the air age at a point, the fresher the air at that point and the better the air quality. It also reflects the room's ability to remove pollutants, with a room with a small average air age being more capable of removing pollutants. Due to the obvious physical significance of air age, it is widely used as an important indicator of the freshness of air in air-conditioned rooms and the ability to change air. Traditionally the air age concept has only considered the interior of the room, the air age at the room inlet is considered to be 0 (100% fresh air). In order to consider the effect of the entire ventilation system including return air, mixed air and flow in the ducts, Tsinghua University has proposed the concept of full air age. The time that the air micro-clusters experience since entering the ventilation system; the air age at the entrance of the room is taken as 0 and the resulting air age is called the room air age. Compared to the room air age, the full air age can be seen as an absolute parameter and the full air ages of different rooms can be compared.

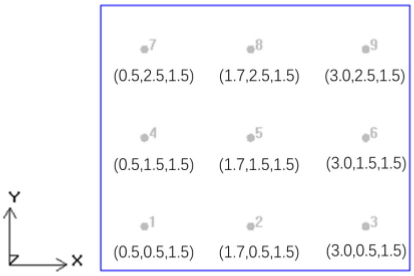
A plot of the air age distribution in the room at a height of 1.5m was obtained by Airpak simulation for different air supply methods, as shown in Figure 3-16.

As can be seen from the figure, Case 3 has a better air replacement effect. The analysis shows that the air age in Case 1 is about 400-640s, with an average value of 585s; in Case 2 the air age is about 190-380s, with an average value of 331s; in Case 3 the air age is between 120-290s, with an average value of 261s. Smaller for mean age of the air coming out near the air conditioner. 9 points on the 1.5m height section are selected for comparison, with the location of the selected points shown in Figure 3-17(a)

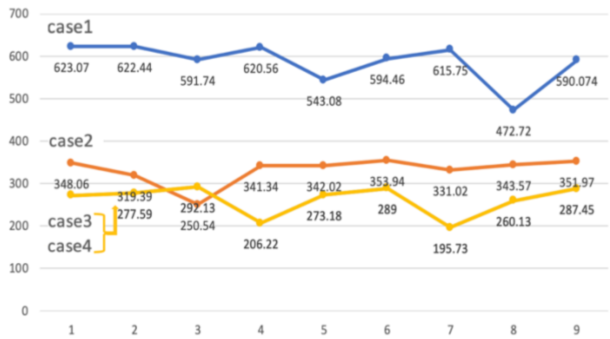



(a) (b) (c)

Figure 3-16 Air mean age at Z=1.5m. (a) Case 1;(b) Case2;(c) Case 3.




(a)




(b)

Figure 3-17 Air mean age at Z=1.5m of case1,2and 3.

Figure 3-17 (b) shows a comparison of the air ages at selected points. It can be seen visually that the average air age for Case 3 is relatively small. Case 1 has a much larger air age than the other two options. According to the definition of air age, the lower the air age, the fresher the air in the room, so the side down and up approach is better than the other two methods.

## 3.7. Conclusion

This chapter establishes a mathematical model and uses FLUENT software to simulate the effect of ventilation on indoor humidity and wall condensation. The simulation indicates that the way of ventilation has a big influence on distribution of condensation on walls surface. Concluded as followed:

The model can realistically simulate the effect of ventilation on the condensation distribution. The ventilation method affects the condensation distribution and condensation sequence of the wall surface, so the wall condensation can be effectively controlled by properly arranging the position of the outlets.

For buildings affected by wet sources, ventilation with lower supply and upper back mode is better than other methods. Case 3 ventilation provides better air flow in the room and better air replacement in terms of air age, which prevents condensation from occurring inside the room.

It can be inferred that when the wall of the ventilated room is added with insulation, replacing the material with good heat insulation and moisture resistance can also control the occurrence of condensation.

However, there are still some important issues that need to be discussed in the future, including the following:

The conclusions of this paper are based on small rooms and it remains to be seen whether similar conclusions can be drawn for other buildings, such as the problem of split-level air conditioning in tall spaces.

For split type air conditioner room, airflow organization distribution with the installation position of the air conditioner, indoor equipment placement and indoor personnel distribution, different airflow organization will form different velocity field and temperature field.

For office buildings, which are now mostly controlled by central air conditioning, the VRV plus fresh air air conditioning system and the primary return air conditioning system are also worth discussing with regard to the distribution of the indoor temperature and humidity fields and the indoor air quality.

# CHAPTER 4
# Analysis of the Effects of Ventilation Method on Indoor Humidity Distribution and Condensation of Single Room by CFD method

# 4. Analysis of the Effects of Ventilation Method on Indoor Humidity Distribution and Condensation of Single Room by CFD method

## 4.1. The summary of CFD simulation

### 4.1.1. Establishment of a basic physical model

This chapter simulated the effects of two different ventilation method and different placement of humidifier on condensation. The simulation is dedicated to single room(5m*5m*2.8m) The room plan is shown in Figure 3-1. There are two type of air conditioning system. The way of A type is air supply and air outlets at the same direction. The B type is the air supply and air outlets on the opposite of the room. A total of 10 placement for the humidifier in the simulation experiment are shown in the Figure 1. In the model, the humidifiers are placed in five locations, two heights and ten places. The letters indicate two different ways of ventilation(A,B). The height is indicated in parentheses. For example, A1(0) and A1(0.5) indicate that the humidifier is placed on the ground at position 1 of the ventilation mode A and placed at the height of 0.5 m from the ground at position 1 of the ventilation mode A.

In the model the wall on the side of window are made by concrete with overall heat transfer coefficient of 1.74W/m2・K. The heat transfer coefficient of the window is 0.63 W/m2・K. The other walls are insulated and desiccated. A humidifier was placed in the room to simulate humidification. The model boundary conditions are set in the Table3-1

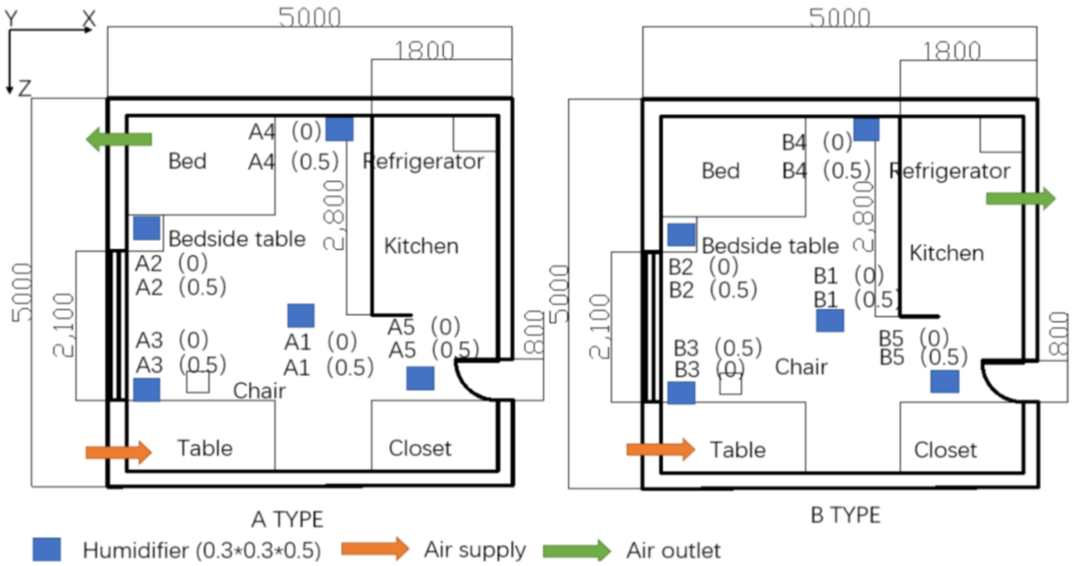



Figure 4-1 Physical Model of Each Case

Table 4-1 Model Boundary Conditions

| Boundary | Parameter Setting |
|---|---|
| Air supply | Air volume： 216m^3/h, Temp.: 30℃， Relative Humidity: 20% |
| Air Outlet | Outflow |
| Wall | Dehumidification， overall heat transfer coefficient 1.74W / (m^2 • K)， Outdoor temperature 5 ℃ |
| Ceiling and Floor | Insulation |
| window | heat transfer coefficient 0.63W / (m^2 • K) |
| Initial environment | Temp. 23℃ RH Φ=20% |
| Penetrating wind | 0.2m/s |
| Humidifier | Amount of moisture 300ml/h |

Table 4-2 Case number and the humidifier position

| | | | |
|---|---|---|---|
| case 1 | no humidifier | | |
| case 2 | A1(0) | case 12 | B1(0) |
| case 3 | A1(0.5) | case 13 | B1(0.5) |
| case 4 | A2(0) | case 14 | B2(0) |
| case 5 | A2(0.5) | case 15 | B2(0.5) |
| case 6 | A3(0) | case 16 | B3(0) |
| case 7 | A3(0.5) | case 17 | B3(0.5) |
| case 8 | A4(0) | case 18 | B4(0) |
| case 9 | A4(0.5) | case 19 | B4(0.5) |
| case 10 | A5(0) | case 20 | B5(0) |
| case 11 | A5(0.5) | case 21 | B5(0.5) |

### 4.1.2. Simplification of the physical model

If the simulated state following the actual state of lab, the error between the actual measurement and the simulation experiment will be minimized. However, due to the accuracy of the model will affect the division of the mesh, the operation of the computer's performance requirements is high, It is difficult for ordinary computers to meet the requirements, so in order to simulate the smooth progress of this model I affect the room airflow organization smaller objects to a certain degree of simplification, thereby reducing the operation of the computer performance requirements. 5 points simplifying assumption were made for the model ：

1.The effects of gravity fields on air and water vapors are ignored;

2.Both air and water vapor are considered to be incompressible fluids with a constant density;

3.When the water vapor evaporates from the water surface, only latent heat exchange is performed,

and sensible heat exchange is not considered;
4.When water vapor condenses on the wall, the heat release ignored from the phase change process;
5.Wall surfaces moisture absorption capacity is not considered during wall condensation.

## 4.2. Mathematical method

In the building indoor environment, generally the water vapor in the humid air is in a superheated state and the content is very low, so at a certain extent, it can be approximated as the ideal gas. At the same time the density is small, the diffusion of wet air can be ignored.

The basic laws of conservation of heat and mass transfer between the fluid and the fluid and between the fluid and the solid interface are: mass conservation law, momentum conservation law, and energy conservation law. The mathematical models that generally describe these conservation laws are the corresponding governing equations, namely the mass continuity equation, the momentum equation, and the energy equation. As follows：
Mass continuity equation：

$$\frac{\partial \rho}{\partial_t} + \nabla \cdot (\rho U) = 0 \qquad (4\text{-}1)$$

In the formula, ρ is the fluid density, kg/$m^3$； U is the gas velocity vector, $U_{x,y,z}$.
In this study, considering air as an incompressible fluid, the continuity equation can be simplified to:

$$\nabla U = 0 \qquad (4\text{-}2)$$

Momentum equation:

$$\frac{\partial(\rho U)}{\partial_t} + \nabla \cdot (\rho U \otimes \mathrm{U}) = -\nabla P + \nabla \cdot \tau + \rho_g \qquad (4\text{-}3)$$

where $\otimes$ is the outer product: $U \otimes \mathrm{V} = \mathrm{U}V^T$,$\rho$ is the fluid density ,U is the flow velocity vector,

$\nabla \cdot$is the divergence ,P is the pressure,t is the time$\tau$ is the derivatoric stress tensor, which has order two,$\tau = \mu(\nabla U + (\nabla U)^T) - \frac{2}{3}\delta\nabla \cdot U$

g represents body accelerations acting on the continuum, for example gravity, inertial accelerations, electrostatic accelerations.
The energy equation：

$$\frac{\partial \rho h_{tot}}{\partial t} - \frac{\partial_p}{\partial t} + \nabla \cdot (\rho U h_{tot}) = \nabla \cdot (\lambda \nabla T) + \nabla \cdot (U \cdot \tau) + U \cdot S_M + S_E \qquad (4\text{-}4)$$

Where：
$h_{tot}$ is the total enthalpy of the fluid, $h_{tot} = h + 1/2U^2$ J/kg.
$h$ is the static enthalpy of the fluid, J/kg. And $\lambda$ is the thermal conductivity of the fluid, $W/(m \cdot K)$.
$S_E$ is the source of fluid energy.

There is no chemical reaction in the numerical simulation process of this subject, and the transport equation of the unreacted material is as shown in the following formula:

$$\frac{\partial(\rho Y_i)}{\partial_t} + \nabla \cdot (\rho v Y_i) = -\nabla J_i \tag{4-5}$$

$Y_i$ is the mass fraction of each substance predicted by FLUENT by the convective diffusion equation of the i$^{th}$ substance; $J_i$ is the diffusion flux of substance i, which is produced by the concentration gradient?

When the moving fluid is laminar:

$$J_i = -\rho D_{i,m} \nabla Y_i \tag{4-6}$$

Where: $D_{i,m}$is the diffusion coefficient of the first species in the mixture.

When the moving fluid is turbulent:

$$J_i = -\left\{\rho D_{i,m} + \frac{\mu_t}{Sc_t}\right\} \nabla Y_i \tag{4-7}$$

Where:$Sc_t$ is the turbulent Schmidt number

In the formula "d" is the absolute humidity. The absolute humidity between the mass of water vapor present in moist air to the mass of dry air. Absolute humidity is normally expressed in kilograms of water vapor per kilogram of dry air.

Absolute humidity expressed by mass:

$$\mathrm{d} = \frac{\mathrm{m_w}}{\mathrm{m_a}} \tag{4-8}$$

Where: d = absolute humidity; $\mathrm{m_w}$= mass of water vapor (kg):

$\mathrm{m_a}$= mass of dry air (kg)

The FLUENT 14.0 turbulence module (Standard k-ε) is used to analyze the flow field characteristics. The component transport module (Species Transport) and the multiphase flow module (Multiphase) simulated the mass transfer phase transition of water vapor in the air. The governing equation is as follows:

$$\frac{\partial(\rho\emptyset)}{\partial_\tau} + div(\rho u\emptyset) = div(\Gamma grad\phi) + s \tag{4-9}$$

Where: $\phi$is a general variable;vΓ is a generalized diffusion coefficient; s is a generalized source term.

Wall condensation conditions: When the absolute humidity $D_{air}$ of the air node around the wall surface is higher than the saturated absolute humidity $D_{wall}$ corresponding to the wall temperature, the wall surface is considered to be condensation.

Conditions for no condensation on the wall: Any time τ, when the absolute humidity $Dair$ of the air node around the wall is less than the saturated absolute humidity $Dwall$ corresponding to the wall temperature, the wall is not considered to be condensation

$$Dwall < Dair \tag{4-10}$$

### 4.2.1. Distribution hypothesis and convergence principle of dependent variable

1. Distribution hypothesis of dependent variable

This topic adopts different distribution hypotheses for different dependent variables. When the grid is sufficiently fine, the variation of the dependent variable between the grids decreases, and the impact of such different assumptions will also be reduced. According to different calculation actual situations, the first-order or second-order welcome style is adopted to ensure the accuracy and stability of the calculation results.

2. The principle of convergence

The convergence criterion for the energy equation is generally taken as: $1 \times 10^{-6}$ (for example: the convergence criterion for temperature); the convergence criterion for the flow equation is generally taken as: $1 \times 10^{-3}$ (for example: the convergence criterion for velocity). The convergence criterion also reflects the accuracy of the simulation calculation.

3. Determination of relaxation factor

In the process of calculating by successive iteration method, the choice of relaxation factor is a more critical link. A properly selected relaxation factor can accelerate convergence. Improper selection may cause calculation oscillation or divergence. As far as the solution of the entire nonlinear problem is concerned, there is no complete theory to judge whether the iterative solution method can obtain a convergent solution. The sub-relaxation iteration method is used in FLUENT software to accelerate convergence. Many scholars have studied the determination of the optimal relaxation factor, but they all aim to directly find the optimal relaxation factor. The selection of such relaxation factors is quite complicated. The common method in engineering is to choose different relaxation factors for trial calculation according to the convergence speed of the iteration process, constantly modify, and gradually find the best, until it is satisfied, then fix it and continue the iteration, in order to achieve the purpose of accelerating convergence.

In the calculations in this chapter, the pressure relaxation factor is between 0.1 and 0.3, the momentum relaxation factor is between 0.1 and 0.5, the temperature and viscosity relaxation factor is between 0.7 and 1.0, and the mass force relaxation factor is 0.1. The relaxation factor of the turbulent dissipation rate is 0.3 to 0.5. Based on the microscopic analysis of the characteristics of air turbulence, the main method is to adopt the indoor zero equation model and use the controlled volume dispersion method to discrete the control equations. The professional software Airpak2.1 introduced by Fluent is used for simulation calculation. Airpak2.1 uses FLUENT5.6.6 CFD solver for calculation.

### 4.2.2. Simplification of the physical model

1.The effects of gravity fields on air and water vapors are ignored;

2.Both air and water vapor are considered to be incompressible fluids with a constant density;

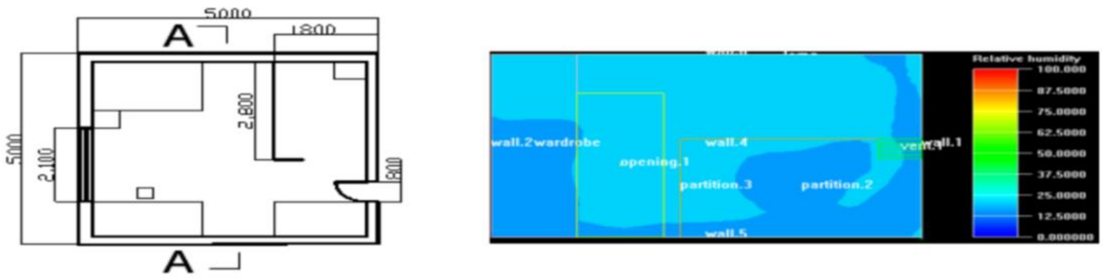
3.When the water vapor evaporates from the water surface, only latent heat exchange is performed, and sensible heat exchange is not considered;

4.When water vapor condenses on the wall, the heat release ignored from the phase change process;
5.Wall surfaces moisture absorption capacity is not considered during wall condensation.

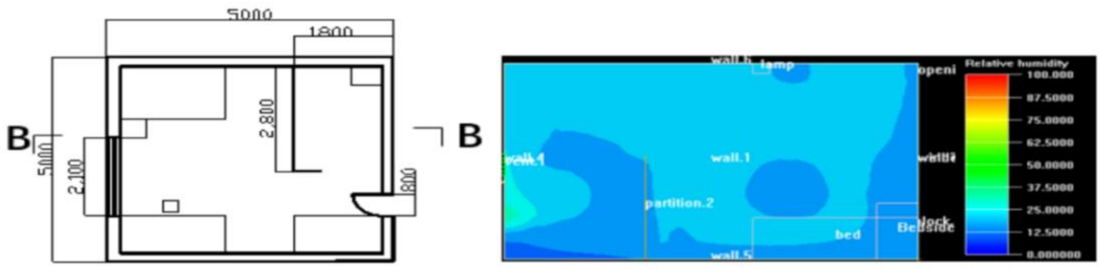
## 4.3. Post-processing and analysis of simulation results

For the first time, the simulation without a humidifier has been completed. The indoor humidity distribution is shown in Figure 4-2. It can be seen that the indoor humidity is about 25% when the humidifier is not used in winter, which is lower than the indoor humidity comfort range This simulation verifies that the indoor humidity range is not in the comfort range without using a humidifier in winter and verifies the need for a humidifier in winter. In addition, it is necessary to study the placement position of the humidifier.

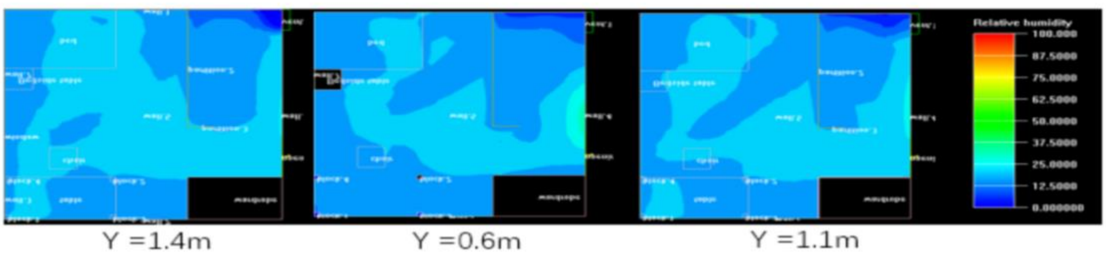
A total of 21 simulations were performed. Figure 4-3 shows the humidity of each simulation case. It can be seen that after using the humidifier, the indoor humidity environment is significantly improved. Under the ventilation mode of B, the indoor humidity is better than the ventilation mode A.




The Humidity Plane Cut of Section A




The Humidity Plane Cut of Section B




The Humidity Plane Cut of Height Section

Figure 4-2 The indoor humidity distribution of Case 1

Figure 4-3 shows the humidity of each simulation case. It can be seen that after using the humidifier, the indoor humidity environment is significantly improved. Under the ventilation mode of B, the indoor humidity is better than the ventilation mode A. In Figure it can be seen that the humidity environment of case 2 is very good, between 40% and 60%. However, in case 12 there are many parts of the humidity above the comfort range.

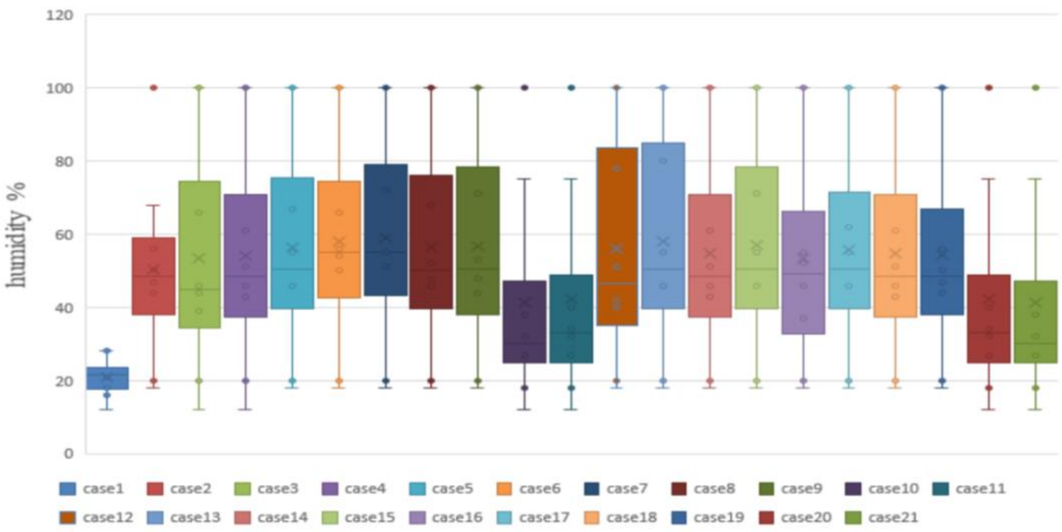
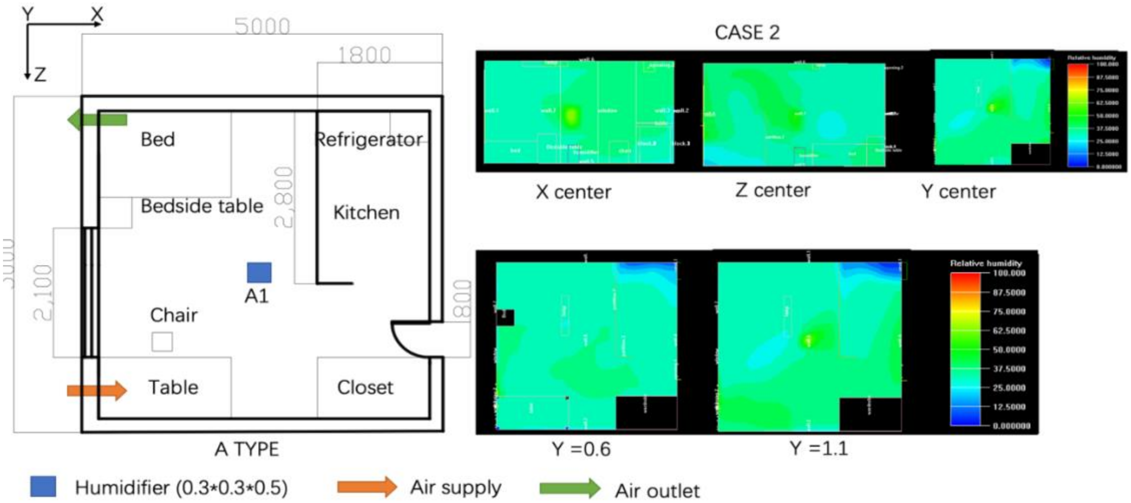
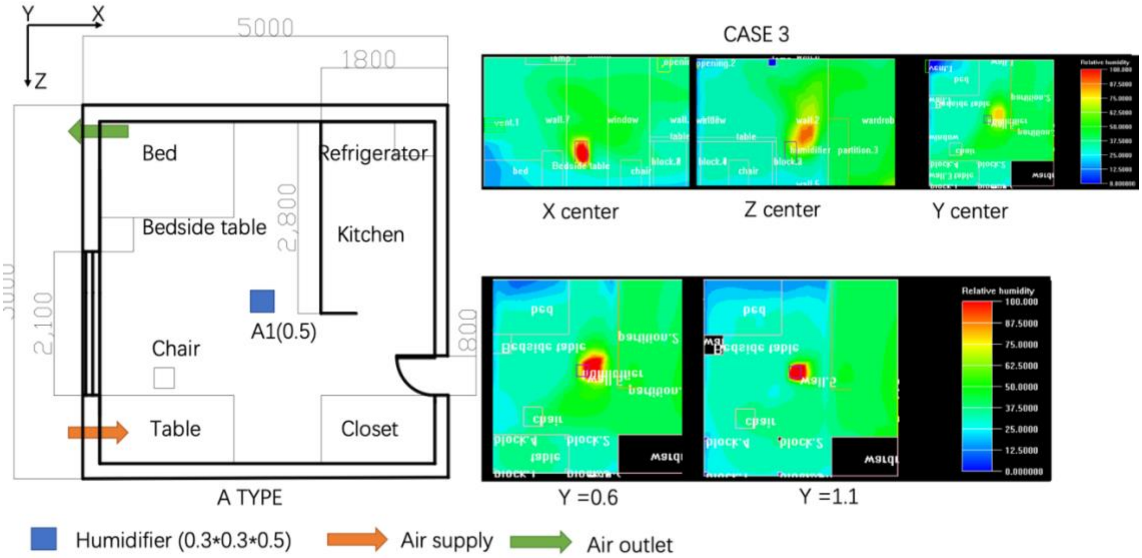
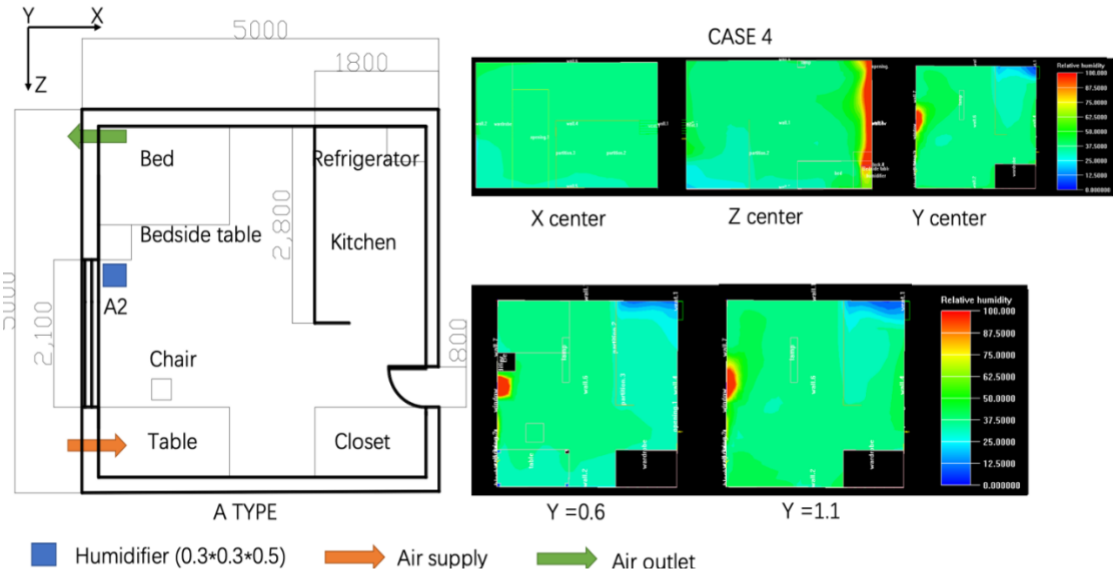


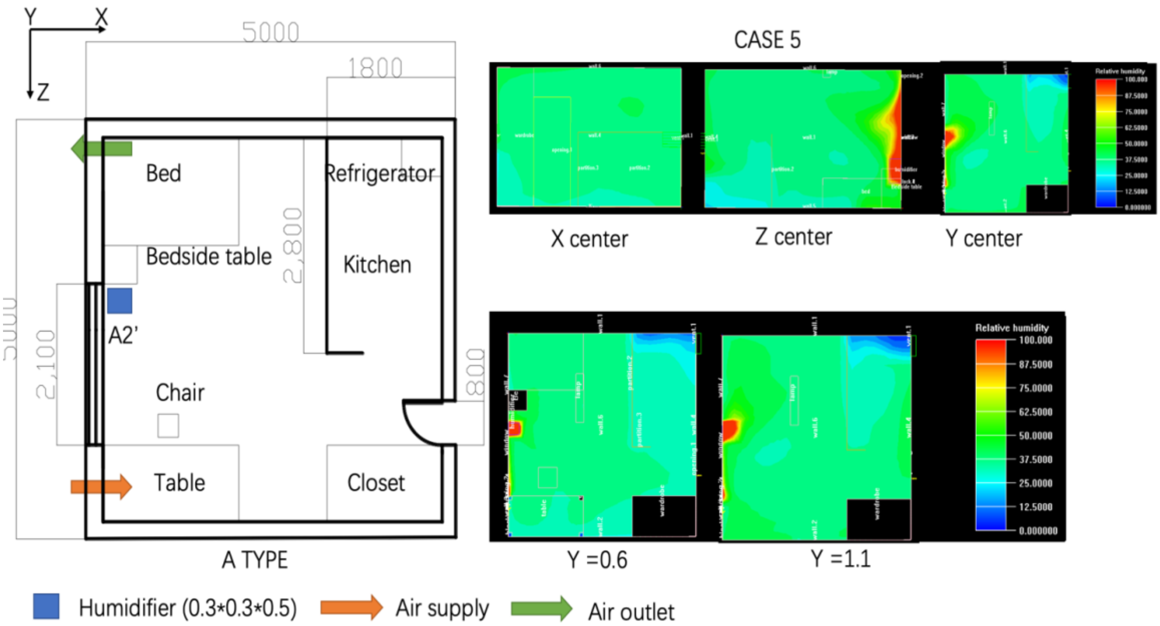

Figure 4-3 The indoor humidity distribution of each cases



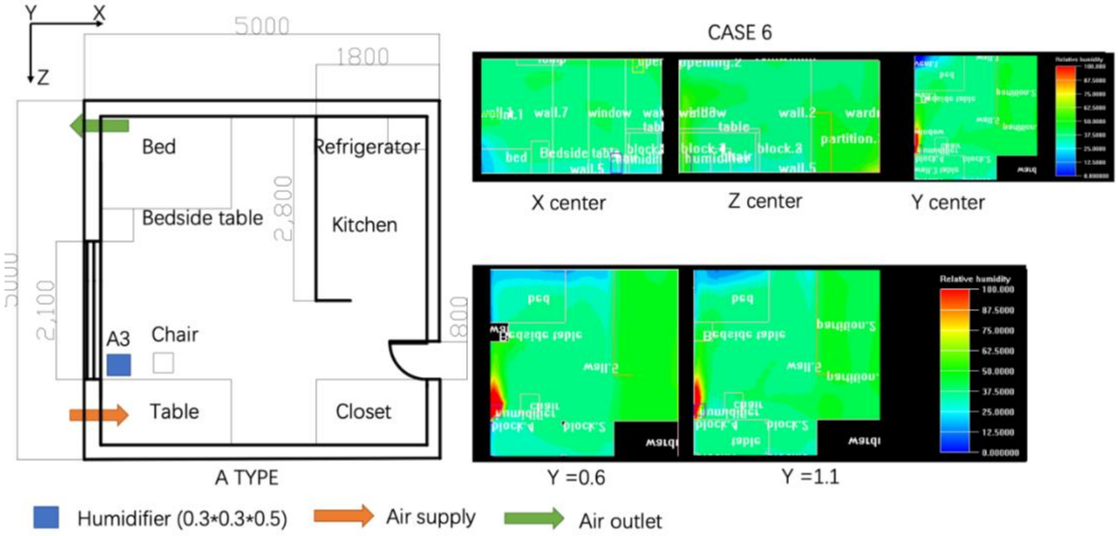



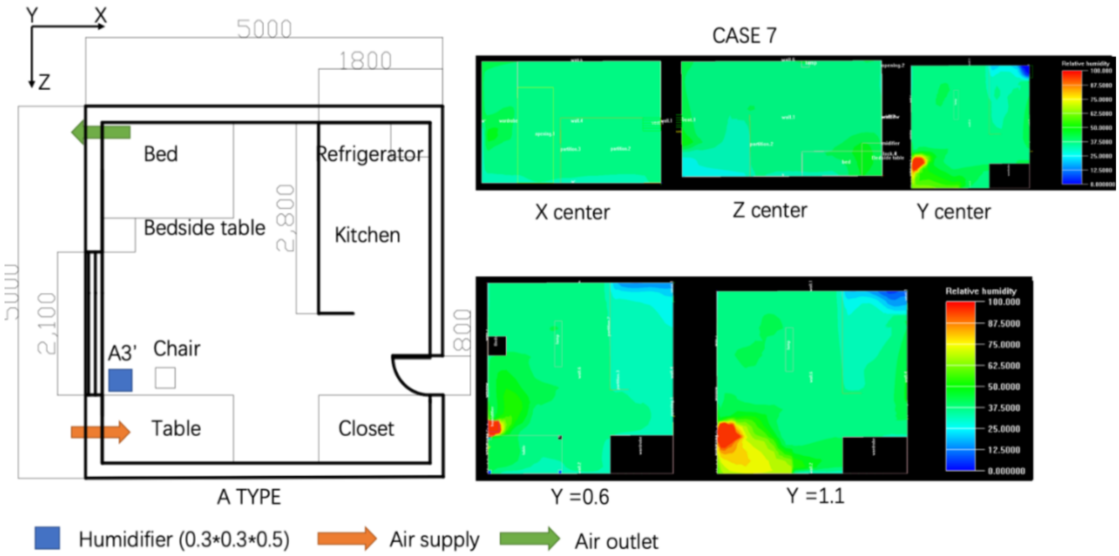
Case 2 and case 3 aim to find out in the same way of ventilation. The humidifier is placed in the middle of the room in case 2 and case 3. The humidity profiles presented in both cases show that the overall humidity in the room is better in case 3 than in case 2. The humidity level in the vicinity of the fridge on the kitchen side is very low due to the partition between the kitchen and the room. However, considering that this location is not the main place for people to stay, it is ignored for the time being. In case 3, there was significant water mist condensation around the humidifier. There is also a low humidity level on the side of the air outlet. The overall average humidity in the room is within a comfortable range at a height cross-section of 1.1m at human sitting height. If the humidifier is placed in the centre of the room, increasing the height of the humidifier will increase the humidity level in the room.



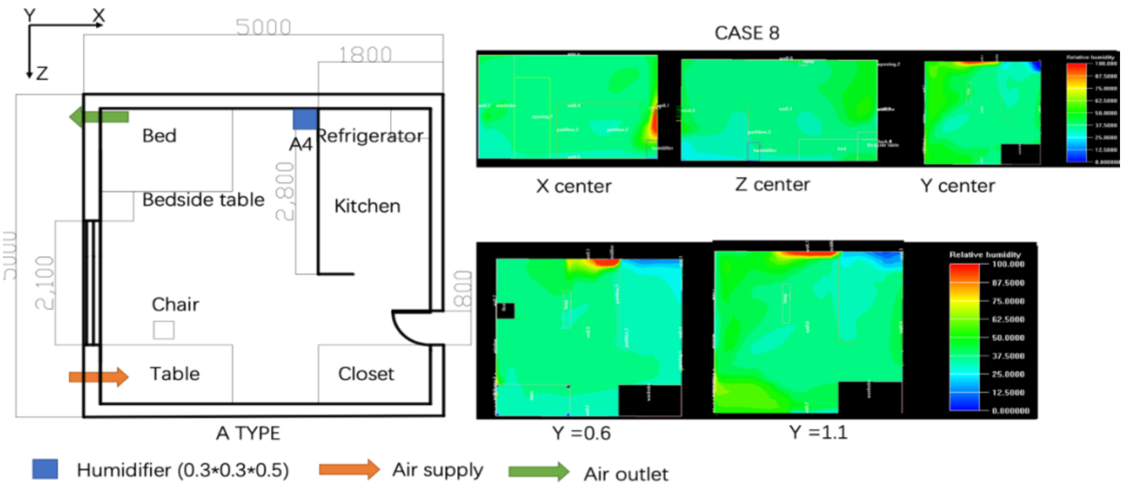



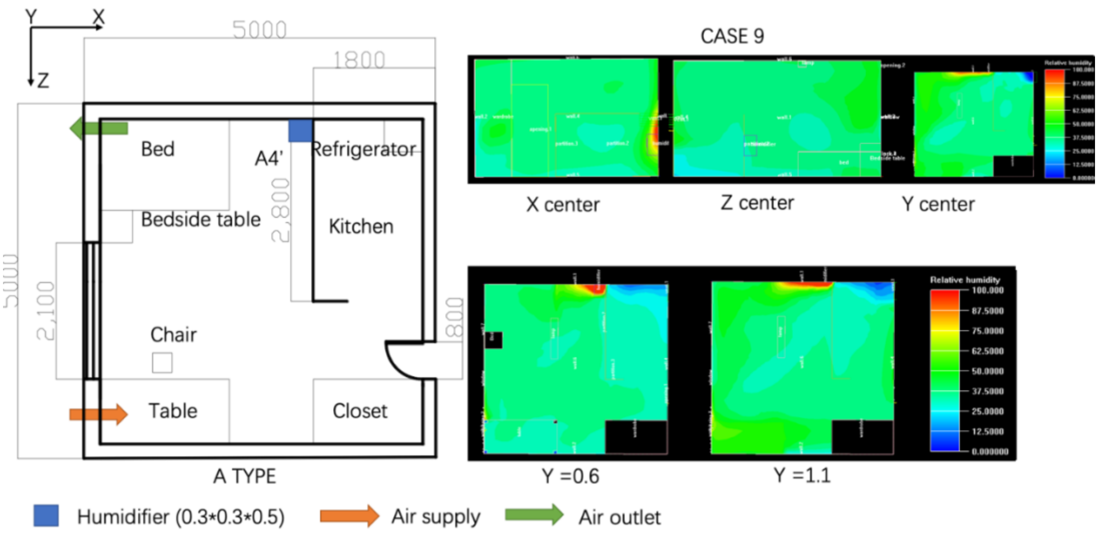
In case 4 and case 5, the humidifier is placed at a window position in the room. The humidity profiles presented in both cases show that the overall humidity in the room in case 4 is not significantly different from that in case 5. The humidity level near the kitchen side of the fridge is still low due to the partition between the kitchen and the room space. However, considering that this location is not the main place for people to stay, it is ignored for the time being. In cases 4 and 5 there is a clear condensation of water mist around the humidifier. As the humidifier is close to the window, condensation can be seen on the window side. At the same time, no low humidity was observed on the side of the air outlet. The overall average humidity in the room is within a comfortable range at a height cross-section of 1.1m at human sitting height. In contrast to case 2 and case 3, where condensation is more likely to occur on the window side and the condensation area becomes larger when the humidifier is placed higher, the humidifier placed on the window side has limited ability to control the overall humidity in the room and is prone to condensation.



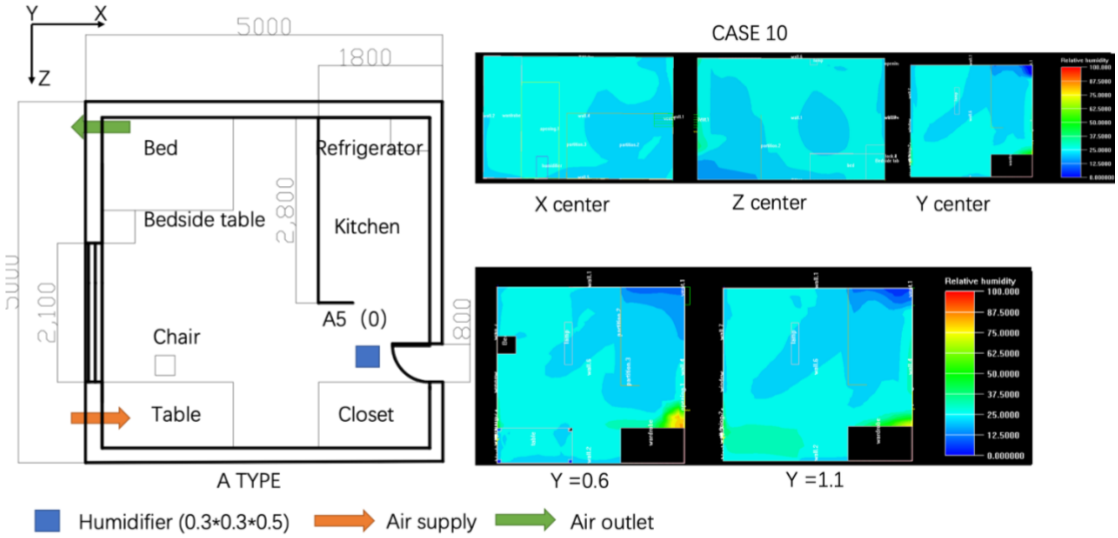



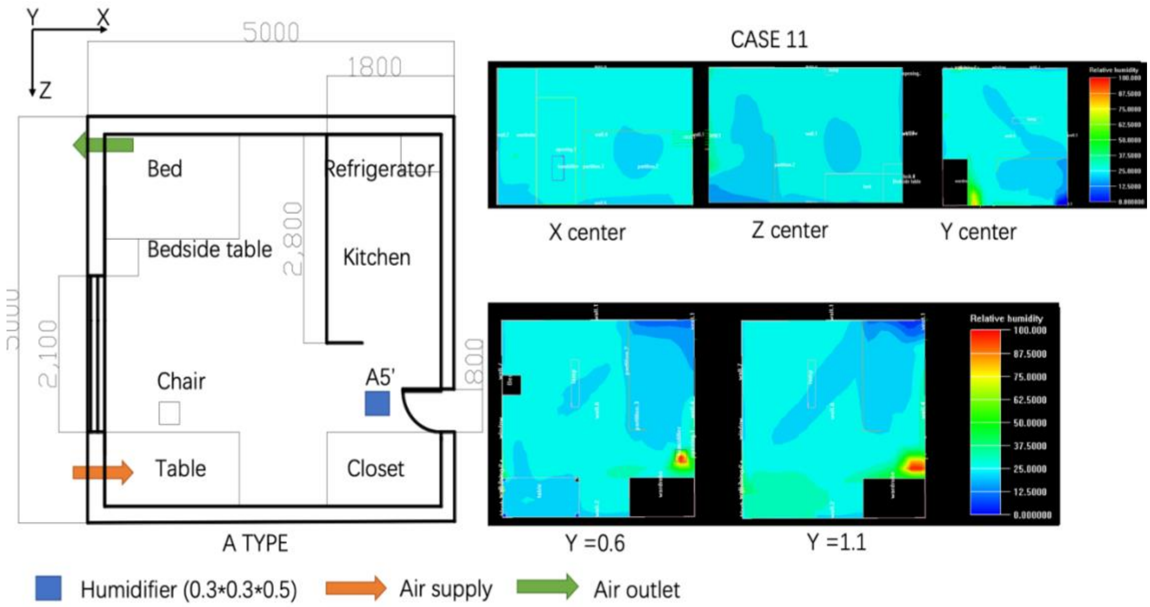
The effect of the humidifier at different heights on the humidity in the room was also investigated in case 6 and case 7, where the humidifier was placed near the window and near the air supply. The humidity profiles presented in both cases show that the overall humidity in the room in case 6 is significantly different from that in case 7. The humidity level near the kitchen side of the fridge is still low due to the partition between the kitchen and the room space. However, considering that this location is not the main place for people to stay, it is ignored for the time being. In cases 6 and 7 there was a noticeable condensation of water mist around the humidifier. However, there was no visible condensation on the window side of the humidifier as it was close to the air outlet, unlike in case 4 and case 5. In case 6, a low humidity level was also observed on the side of the air outlet. The possible influencing factors are the distance from the humidifier and the fact that the humidifier is located too close to the air outlet, which is affected by the air convection. The overall average humidity in the room is within a comfortable range at a height of 1.1m at human sitting height. Compared to the previous case, the condensation on the window side is improved by placing the humidifier near the air outlet, even though it is also located on the window side. The low humidity on the bed side was also significantly improved when the humidifier was placed higher, so the humidity in the room changed significantly when the humidifier was placed on the window side and close to the air outlet, and the low humidity on the return side was alleviated when the humidifier was raised.



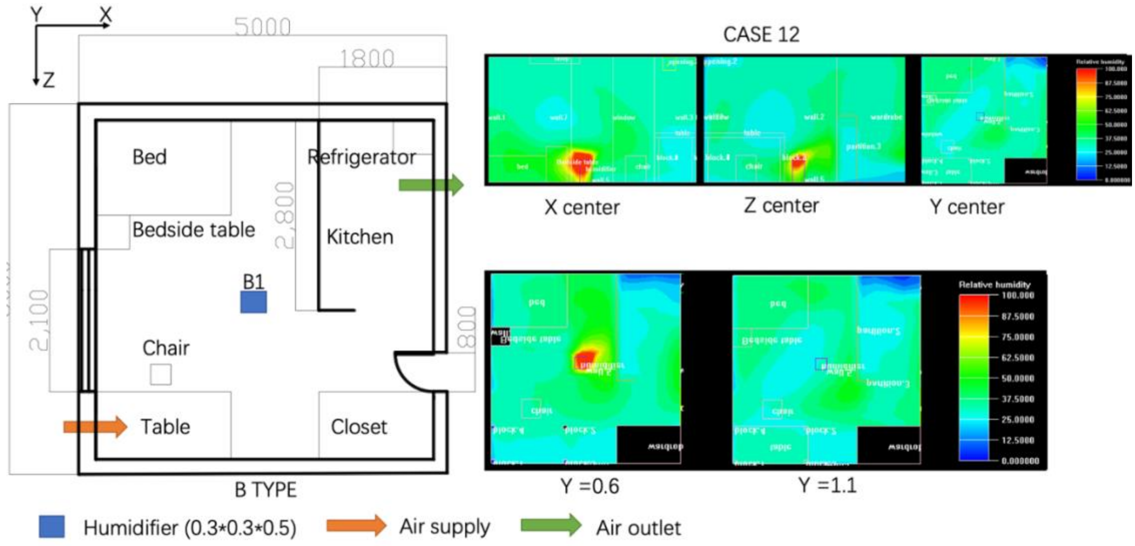



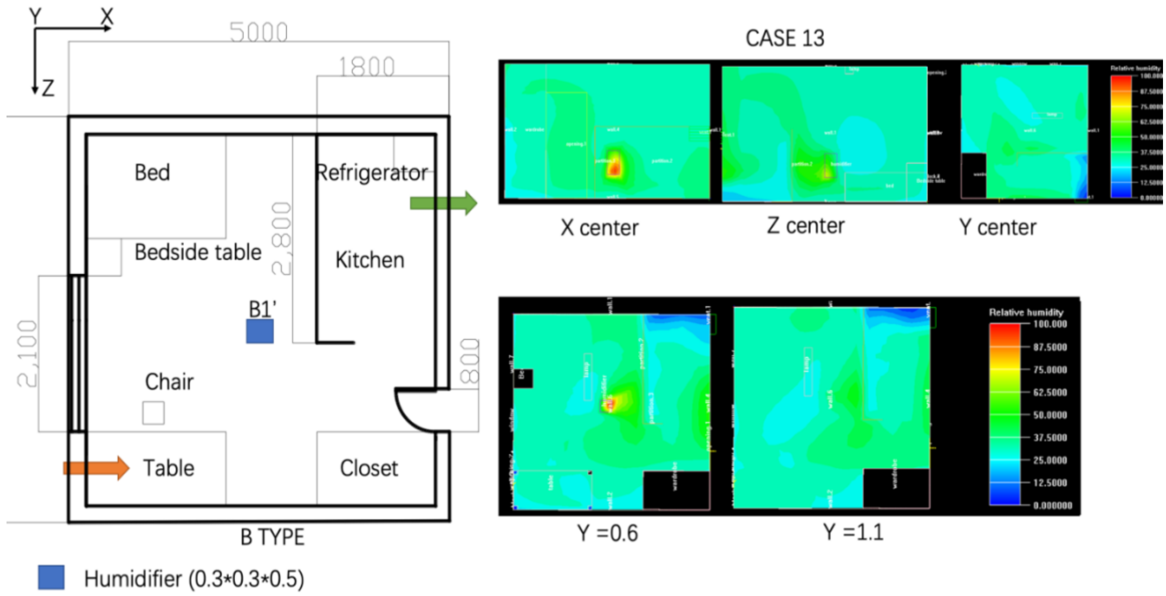
Due to the low humidity level in the kitchen, the humidifier was placed near the external partition of the kitchen and the effect of the humidifier at different heights was investigated in cases 8 and 9 with the same ventilation. The humidity profiles presented in both cases show that the overall humidity in the room in case 8 is not significantly different from that in case 9. However, due to the partitioning of the kitchen space from the room, the humidity level near the refrigerator on the kitchen side still did not improve. However, considering that this location is not the main place for people to stay, it is ignored for the time being. In cases 8 and 9 there was a noticeable condensation of water mist around the humidifier. As the humidifier is close to the wall, a small area of visible water mist condensation was found on the wall in case 9. The low humidity level on the outlet side was also improved. The overall average humidity in the room is within a comfortable range at a height of 1.1m at human sitting height. In comparison to the previous case, the humidity distribution in the room is good with the humidifier placed here and at a higher level.



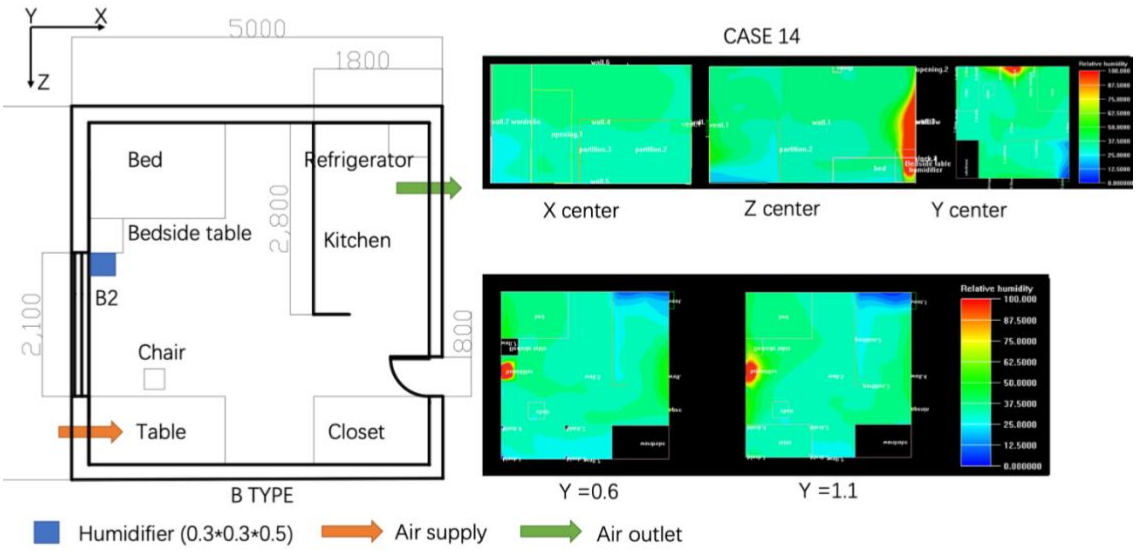



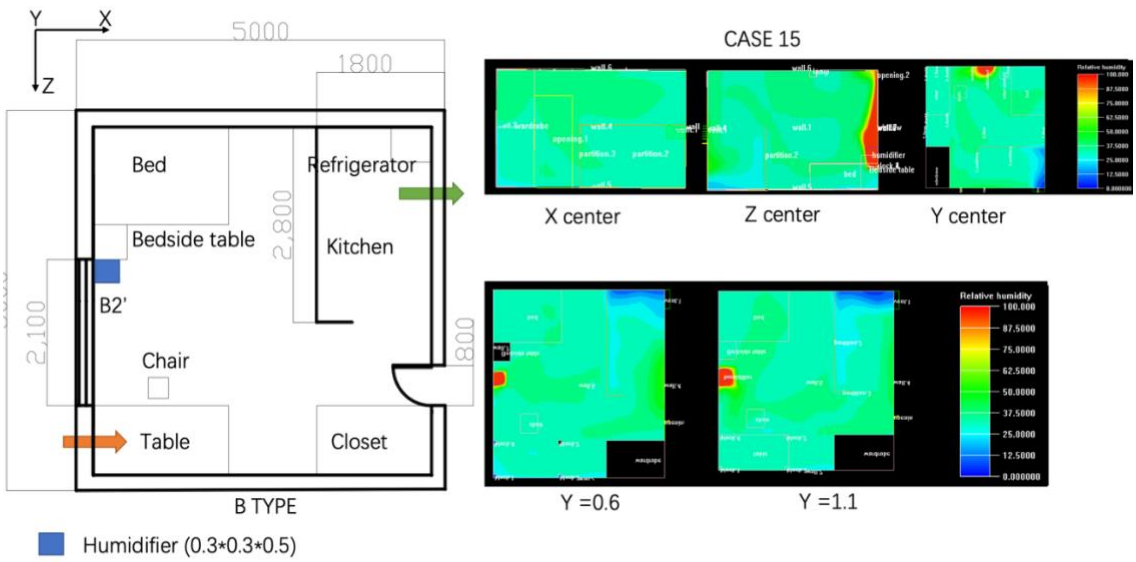
Case 10 and case 11 show the humidity distribution with the same air supply as before, with the humidifier placed at the door. It is clear that the overall humidity distribution in case 10 and case 11 is below the humidity range to which the human body is adapted. The placement of the humidifier does not improve the humidity in the room. The humidity in the room is low near the floor under the effect of the air conditioner, but after raising the humidifier, we can see in case 11 that there is a clear condensation of water mist around the humidifier, but this does not have a significant effect on the overall humidity in the room. In conclusion, when the room is supplied with air from the same side, the humidifier can be placed at a certain height around the air inlet to control the overall humidity of the room better and to prevent the occurrence of condensation on the window side to a great extent.



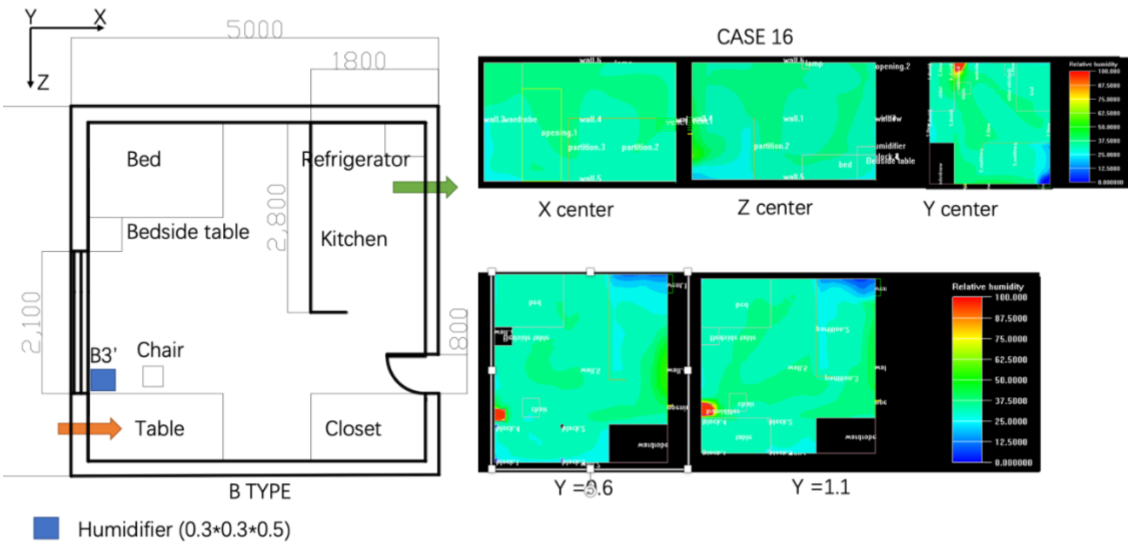



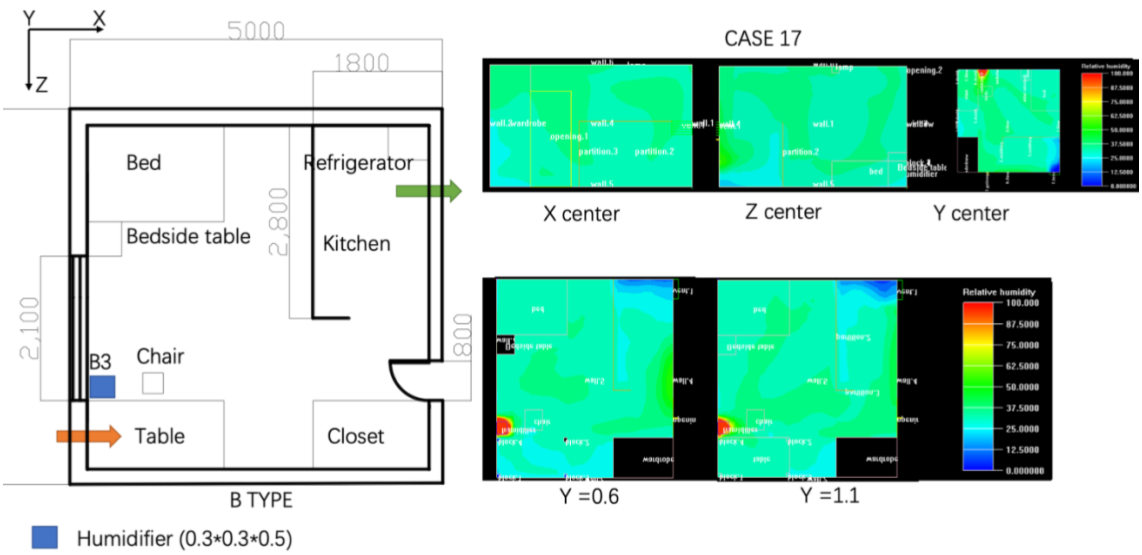
After changing the air supply and return, we repositioned the humidifiers in the original ten locations and explored the humidity distribution in the room at different heights of the humidifiers. case12 and case 13 had the humidifiers placed in the centre of the room. case 12 had the humidifiers placed on the floor and case 13 had the humidifiers placed at a height of 0.5m above the floor. The humidity graph shows that the humidity in the room is at a comfortable level for the human body. As can be seen from the graphs, in case 12 there is a large range of water mist condensation around the humidifier. The humidity is also at a comfortable level for the human body at a working height of 1.1m. The area of water mist condensation around the humidifier decreases when the humidifier is raised due to the airflow organisation in the room. Due to the presence of a partition between the kitchen space and the main space, the humidity level is lower near the refrigerator, but can be considered ignored as it is not the main activity area. From these two examples it can be seen that raising the humidifier can result in a more even distribution of humidity in the room.



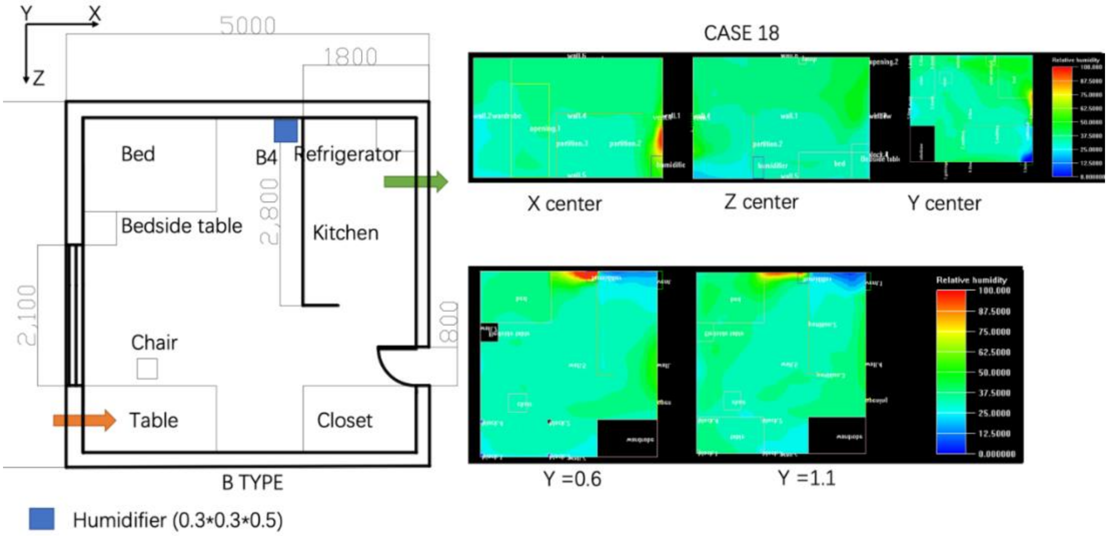



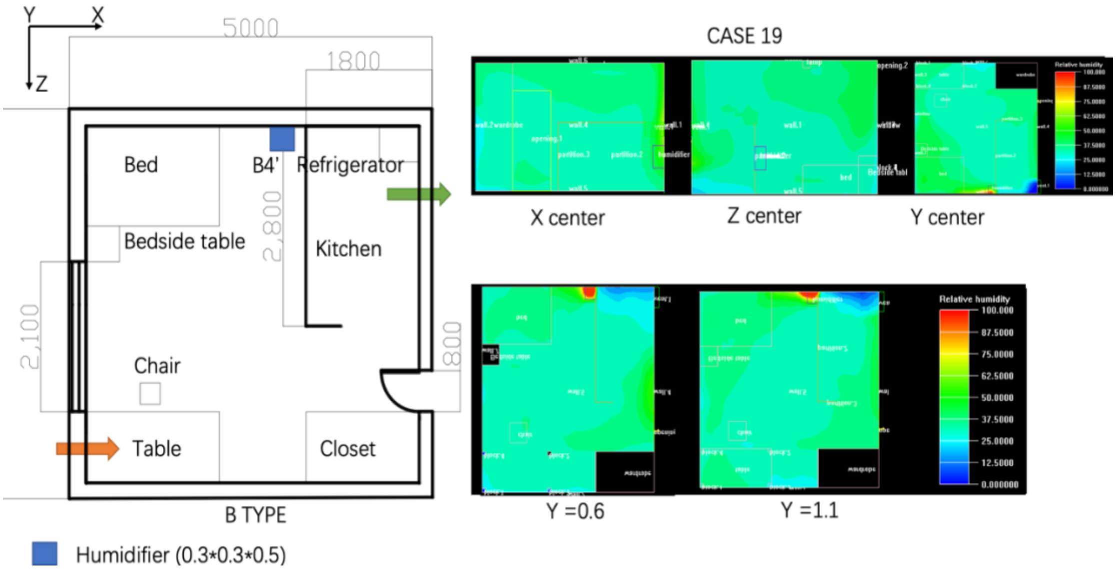
After changing the air supply and return, we repositioned the humidifier at the original ten locations and explored the humidity distribution in the room at different heights of the humidifier. case 14 and case 15 have the humidifier placed by the window on the side of the air supply. case 14 has the humidifier placed on the floor and case 15 has the humidifier placed at a height of 0.5m above the floor. The humidity graph shows that the humidity in the room is at a comfortable level for the human body. As can be seen from the graphs, in case 14 there is a large range of water mist condensation around the humidifier. The humidity is also at a comfortable level for the human body at a working height of 1.1m. However, the close proximity of the humidifier to the window resulted in a very visible condensation of water mist on the window side, which is very likely to have caused condensation on the window, and the area of water mist on the window side was not significantly reduced when the humidifier was raised.



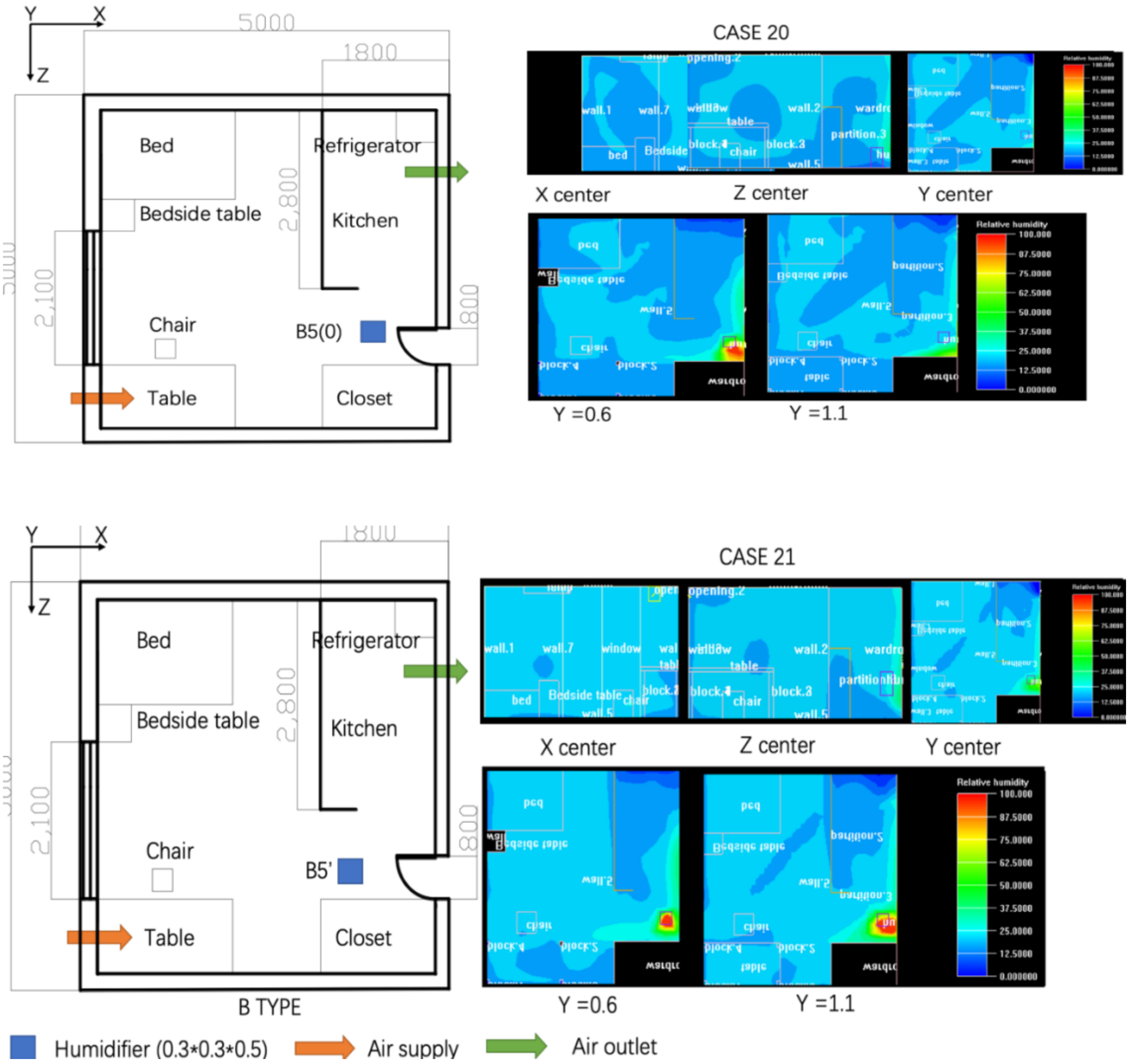



After changing the air supply and return, we repositioned the humidifiers in the original ten locations and explored the humidity distribution in the room at different heights of the humidifiers. case 16 and case 17 had the humidifiers placed by the side windows of the air supply but closer to the air supply than case 15. case 16 had the humidifiers placed on the floor and case 17 had the humidifiers placed at a height of 0.5m above the floor. 0.5m above the floor. The humidity graph shows that the humidity in the room is at a comfortable level for the human body. The humidity distribution in the room is significantly different from that in case 13 and case 15. case 16 and case 17 are closer to the air inlet and do not have a large accumulation of water mist on the window side due to the airflow organization of the air supply. At the exit of the humidifier there is still a visible water mist. The kitchen still does not reach a comfortable humidity range due to the presence of the partition. The humidifier has little effect on the humidity distribution in the room when raised.

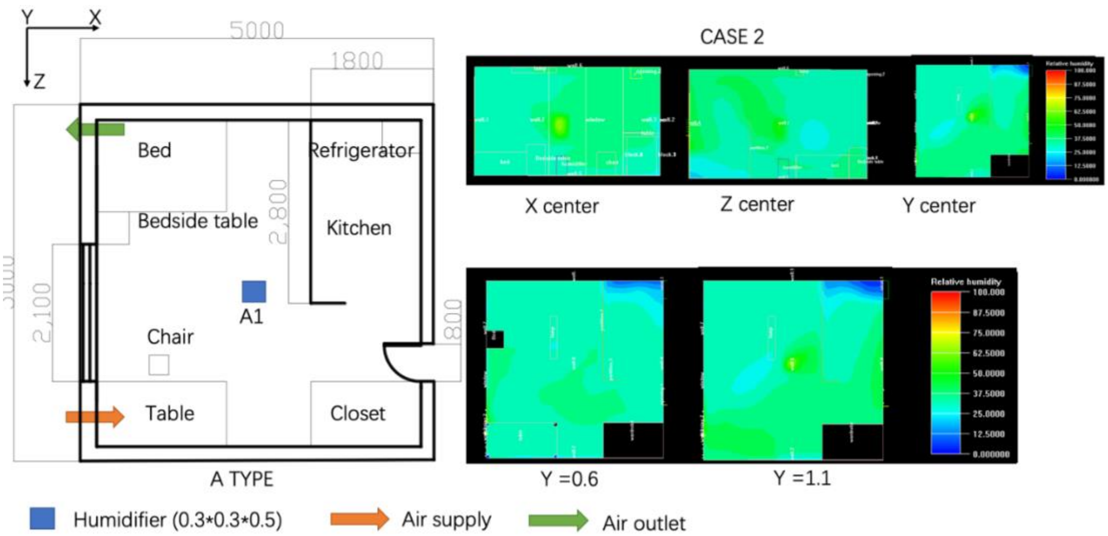


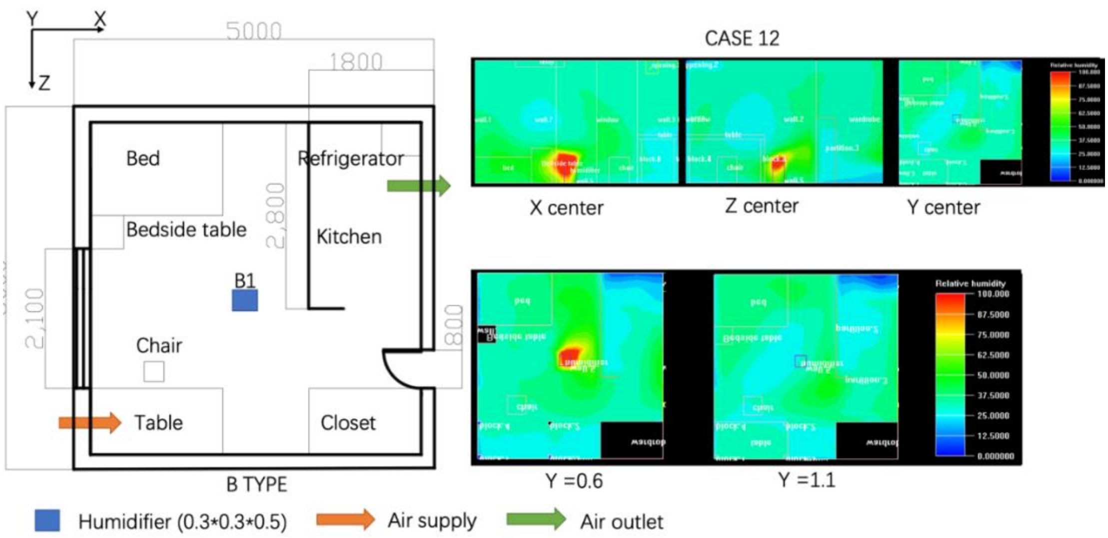




After changing the air supply and return, we repositioned the humidifier at the original ten locations and explored the humidity distribution in the room at different heights of the humidifier. case 18 and case 19 the humidifier was placed near the wall on the other side of the room. case 18 the humidifier was placed on the floor and case 19 the humidifier was placed at a height of 0.5m above the floor. The difference between the two solutions can be seen from the humidity diagrams: in case 18 there is a noticeable collection of water mist on the wall side, but after raising the humidifier, there is no noticeable collection of water mist on the wall side. Only a small area of water mist coalesces at the humidifier outlet. It can be concluded that raising the humidifier has an inhibiting effect on the condensation on the walls of the room.

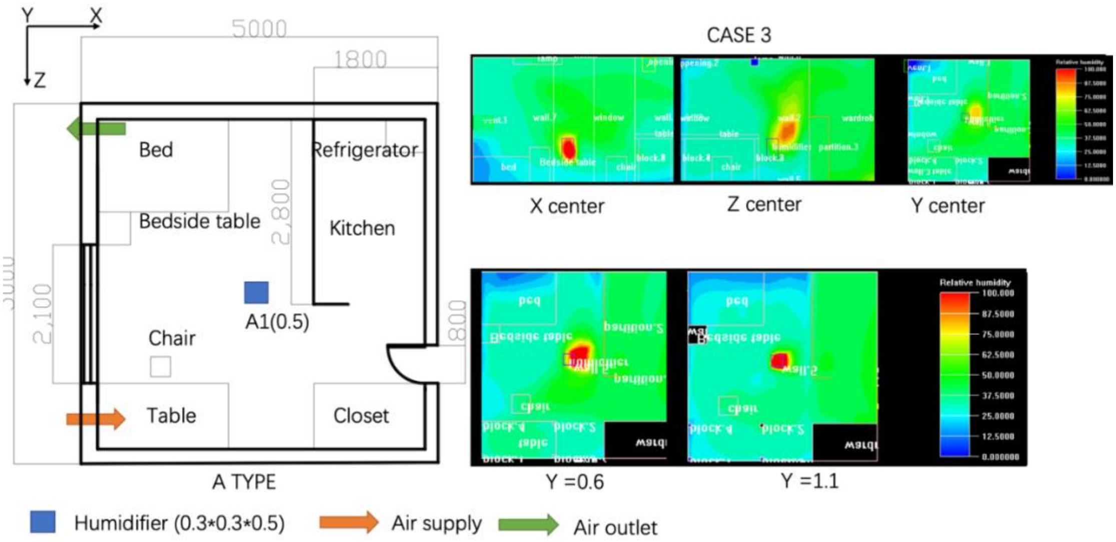


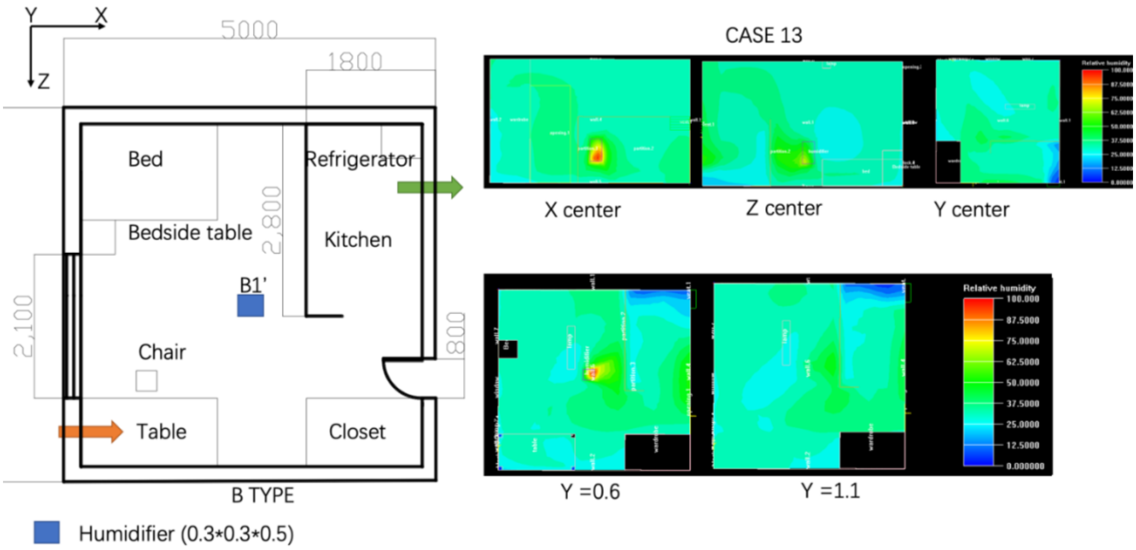

Case 20 and case 21 show the humidity distribution with the same air supply as before, with the humidifier placed at the door. It is clear that the overall humidity distribution in case 20 and case 21 is below the humidity range to which the human body is adapted. The placement of the humidifier does not improve the humidity in the room. The humidity in the room is low near the floor under the effect of the air conditioner, but after raising the humidifier, we can see in case 21 that there is a clear condensation of water mist around the humidifier, but this does not have a significant effect on the overall humidity in the room. In conclusion, when the room is supplied with air from the same side, the humidifier can be placed at a certain height around the air inlet to control the overall humidity of the room better and to prevent the occurrence of condensation on the window side to a great extent.



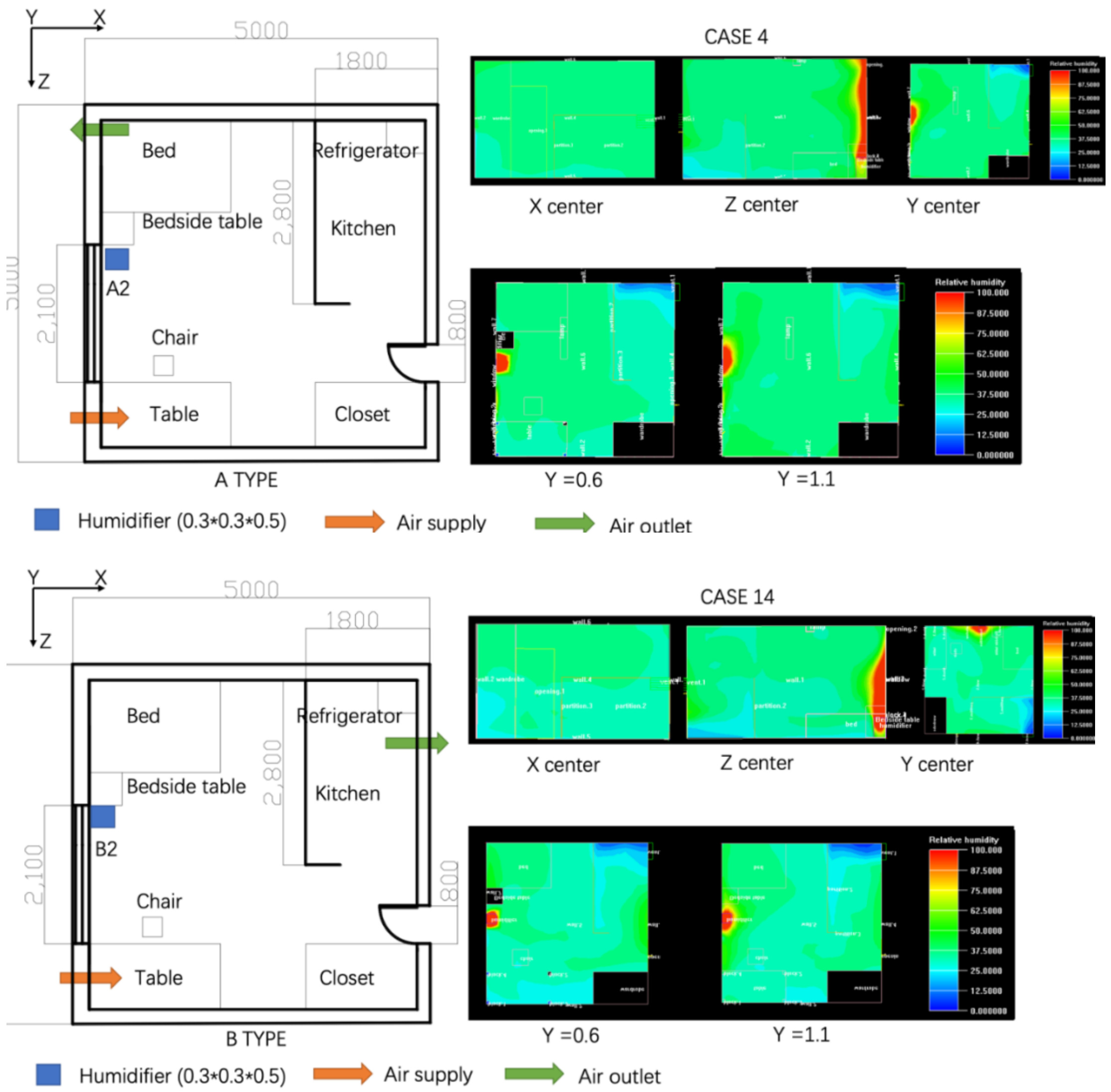




The diagram explores the effect of ventilation on the humidity distribution in the room for different air supply methods. case 2 and case 12 show the humidifier placed on the floor in the middle of the room. In case 12 there is a visible condensation of water mist near the humidifier outlet. In terms of the overall humidity distribution in the room, the average humidity in case 12 is slightly higher than in case 2. Inevitably, the humidity in the kitchen side is still below the average range. No condensation is evident on the windows or walls. In terms of the global humidity range, the heterogeneous air supply and return is more favourable than the homogeneous air supply and return. However, in terms of humidity homogeneity, the same side of the return air supply has no water mist condensation at the humidifier outlet. The air mix is better.

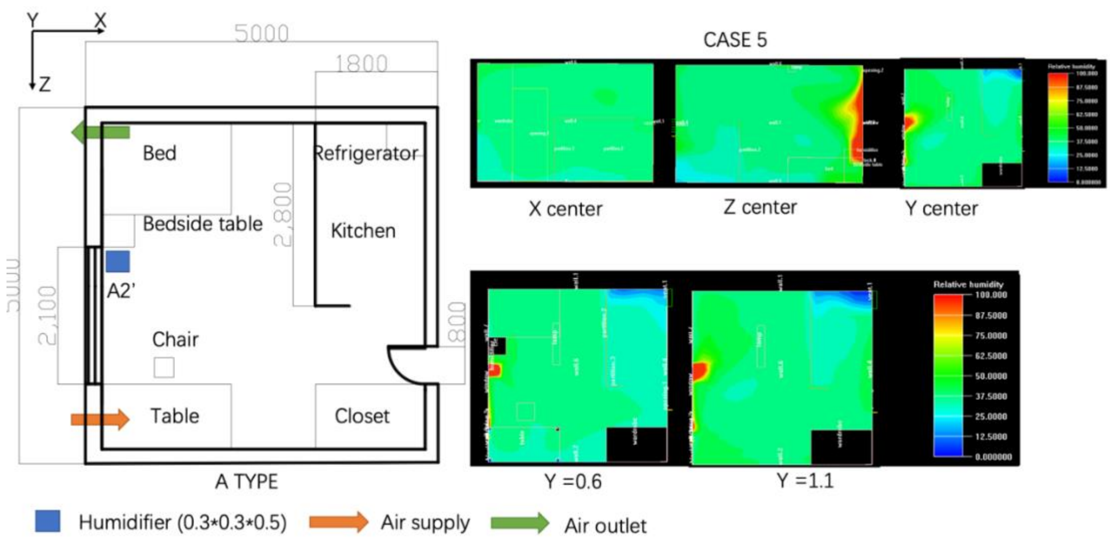


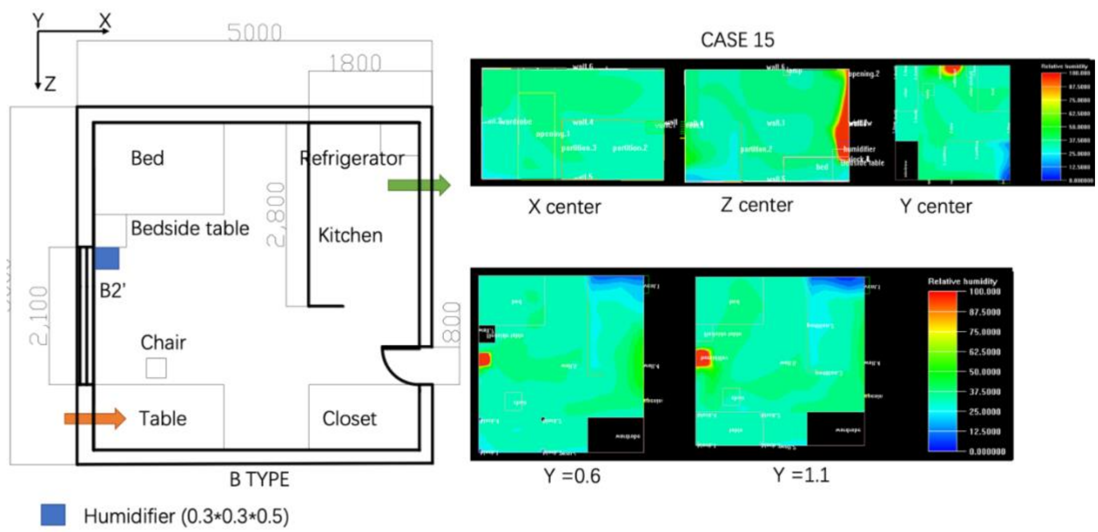




The diagram explores the effect of ventilation on the humidity distribution in a room with different air supply methods. Cases 3 and 13 show the humidifier placed in the middle of the room at a height of 0.5 m above the floor. In Case 13, there is a visible condensation of water mist near the humidifier outlet. In terms of the overall humidity distribution in the room, the average humidity in case 13 is slightly higher than in case 3. Inevitably, the humidity on the kitchen side remained below the average range. No condensation was evident on the windows or walls. In terms of the global humidity range, heterogeneous supply and return air is more favourable than homogeneous supply and return air. However, as far as humidity homogeneity is concerned, the return air on the same side has no water mist condensation at the humidifier outlet. Air mixing is better.

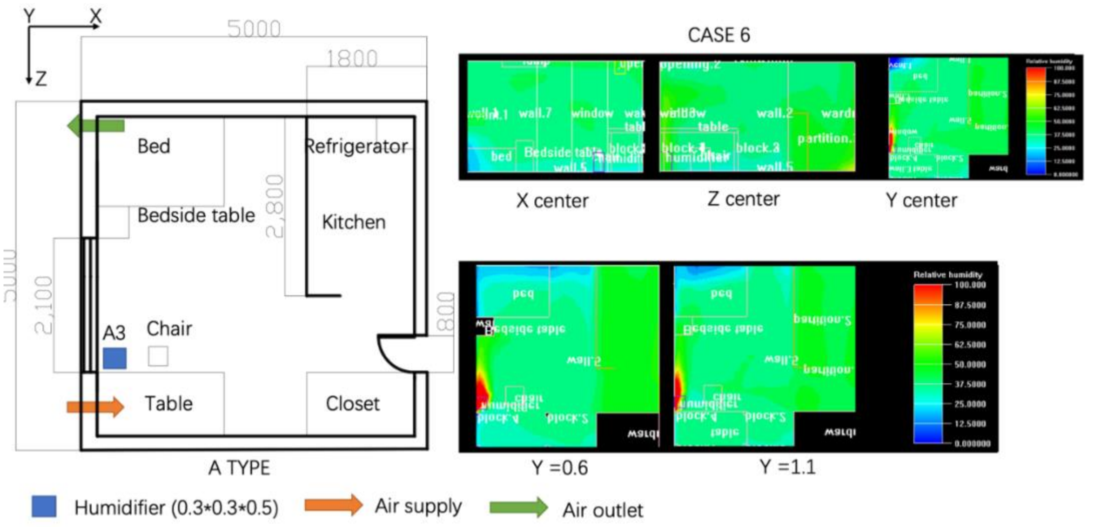


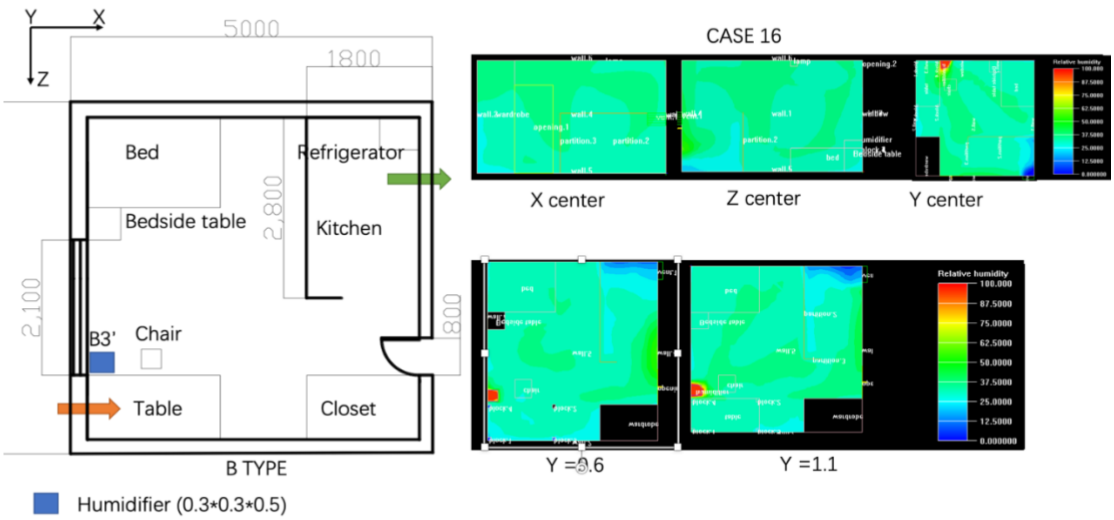

The diagram explores the effect of ventilation on the humidity distribution in a room with different air supply methods. Cases 4 and 14 show the humidifier placed in the room against the window. In the case, there is significant condensation of water mist near the humidifier outlet. In terms of the overall humidity distribution in the room, the average humidity in case 4 is slightly higher than in case 14. Inevitably, the humidity on the kitchen side remained below the average range. As the humidifier is placed on the window side, in cases there is a large range of water mist condensation around the humidifier. The humidity is also at a comfortable level for the human body at a working height of 1.1m. However, the close proximity of the humidifier to the window resulted in a very visible condensation of water mist on the window side, which is very likely to have caused condensation on the window, and the area of water mist on the window side was not significantly reduced when the air supply changed. The condensation area on the window side of case 4 is slightly larger than in case 14, which means that the ventilation in case 14 mixes the room air better and to some extent inhibits the exposure phenomenon.



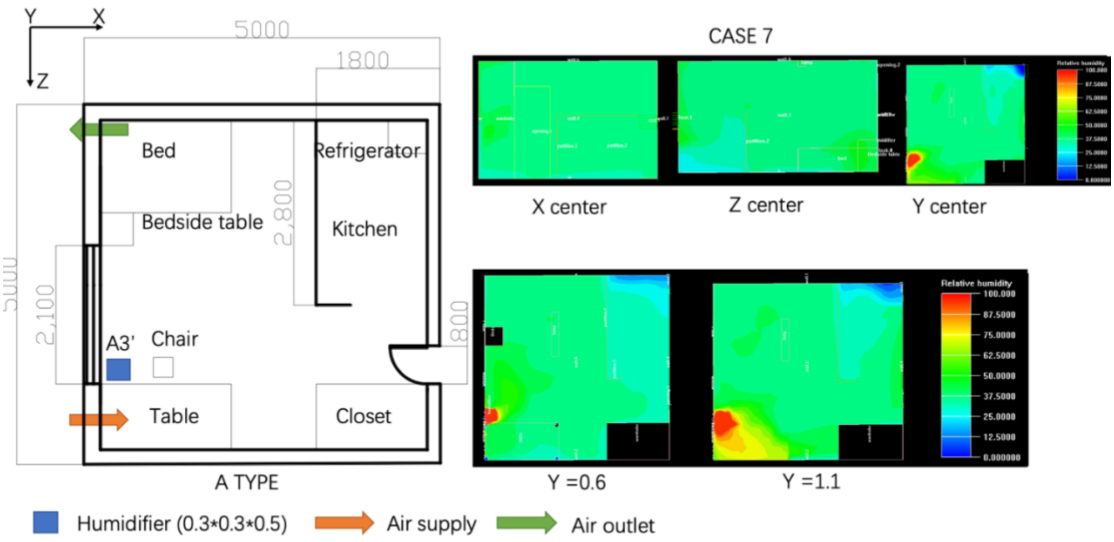



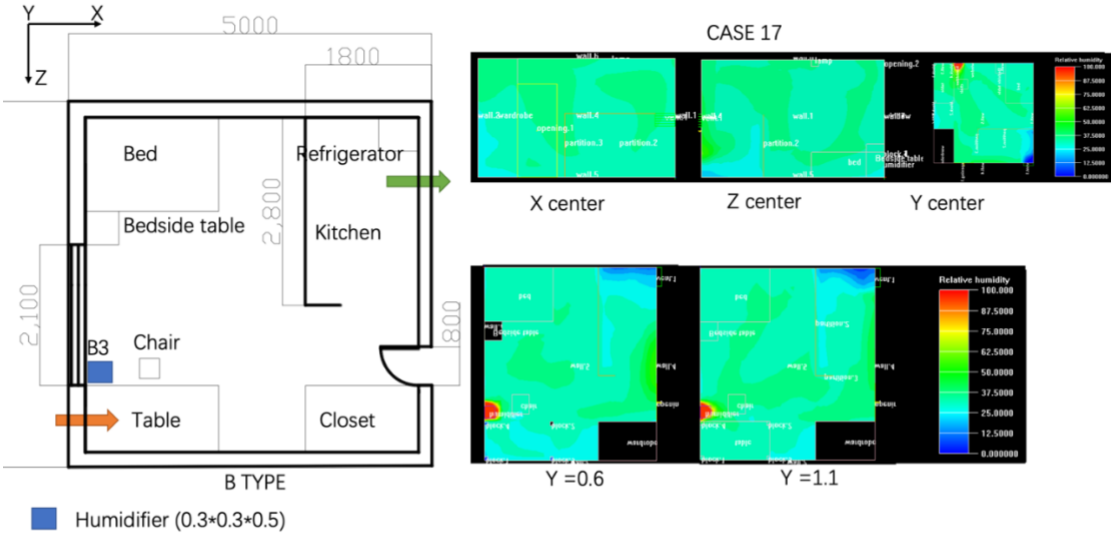

The diagram explores the effect of ventilation on the humidity distribution in a room with different air supply methods. Cases 5 and 15 show the humidifier placed in the room against the window at a height of 0.5 m above the floor.. In the case, there is significant condensation of water mist near the humidifier outlet. In terms of the overall humidity distribution in the room, the average humidity in case 5 is slightly higher than in case 15. Inevitably, the humidity on the kitchen side remained below the average range. As the humidifier is placed on the window side, in cases there is a large range of water mist condensation around the humidifier. The humidity is also at a comfortable level for the human body at a working height of 1.1m. However, the close proximity of the humidifier to the window resulted in a very visible condensation of water mist on the window side, which is very likely to have caused condensation on the window, and the area of water mist on the window side was not significantly reduced when the air supply changed. The condensation area on the window side of case 5 is slightly larger than in case 15, which means that the ventilation in case 14 mixes the room air better and to some extent inhibits the exposure phenomenon.



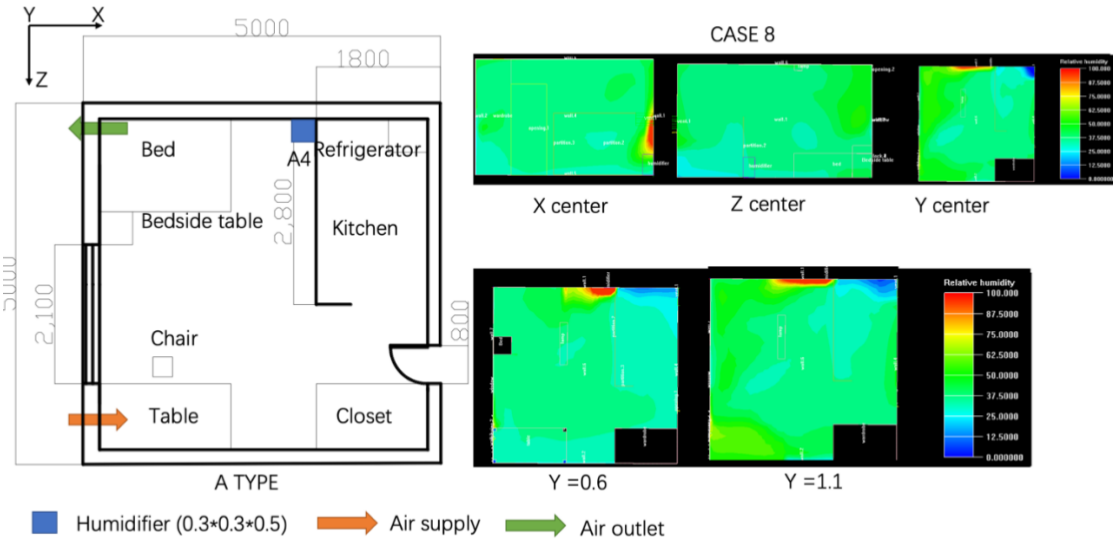



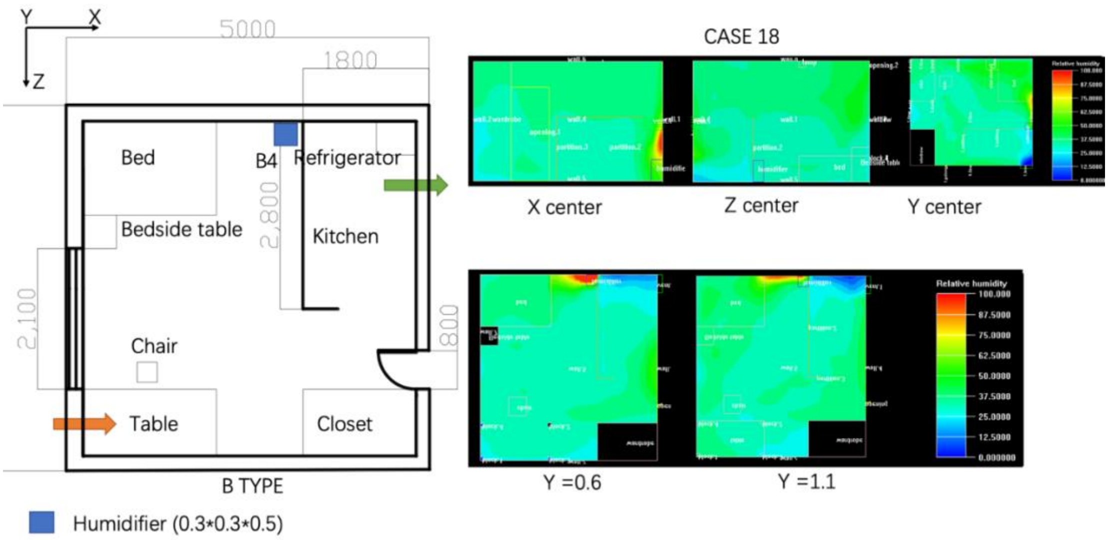

The diagram explores the effect of ventilation on the humidity distribution in a room with different air supply methods. Cases 6 and 16 show the humidifier placed in the room against the window. In the case, there is significant condensation of water mist near the humidifier outlet. In terms of the overall humidity distribution in the room, the average humidity in case 6 is slightly higher than in case 16. Inevitably, the humidity on the kitchen side remained below the average range. As the humidifier is placed on the window side, in cases there is a large range of water mist condensation around the humidifier. The humidity is also at a comfortable level for the human body at a working height of 1.1m. However, the close proximity of the humidifier to the window resulted in a very visible condensation of water mist on the window side, which is very likely to have caused condensation on the window, and the area of water mist on the window side was not significantly reduced when the air supply changed. The condensation area on the window side of case 6is slightly larger than in case 16, which means that the ventilation in case 16 mixes the room air better and to some extent inhibits the exposure phenomenon.



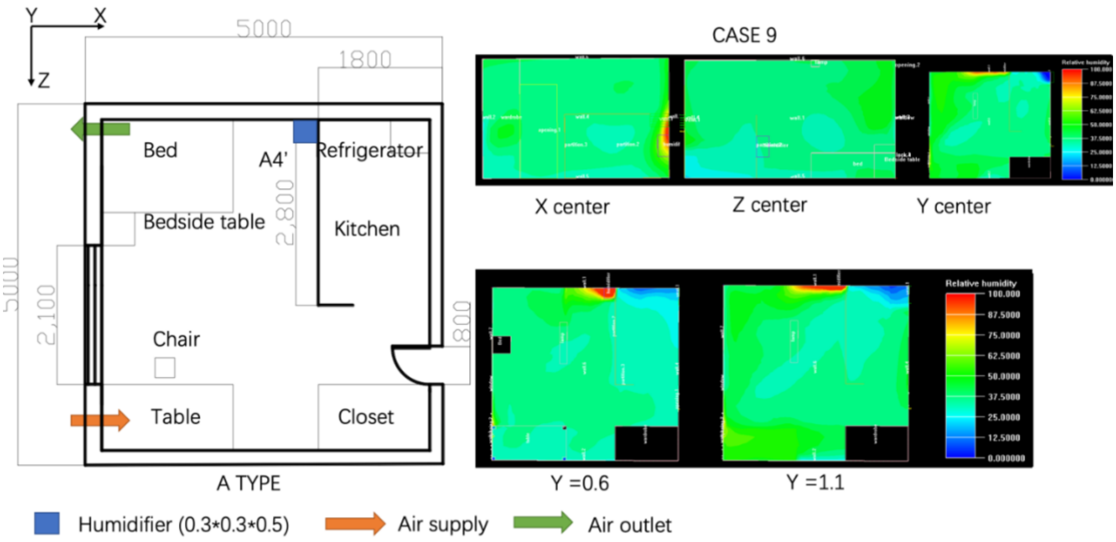



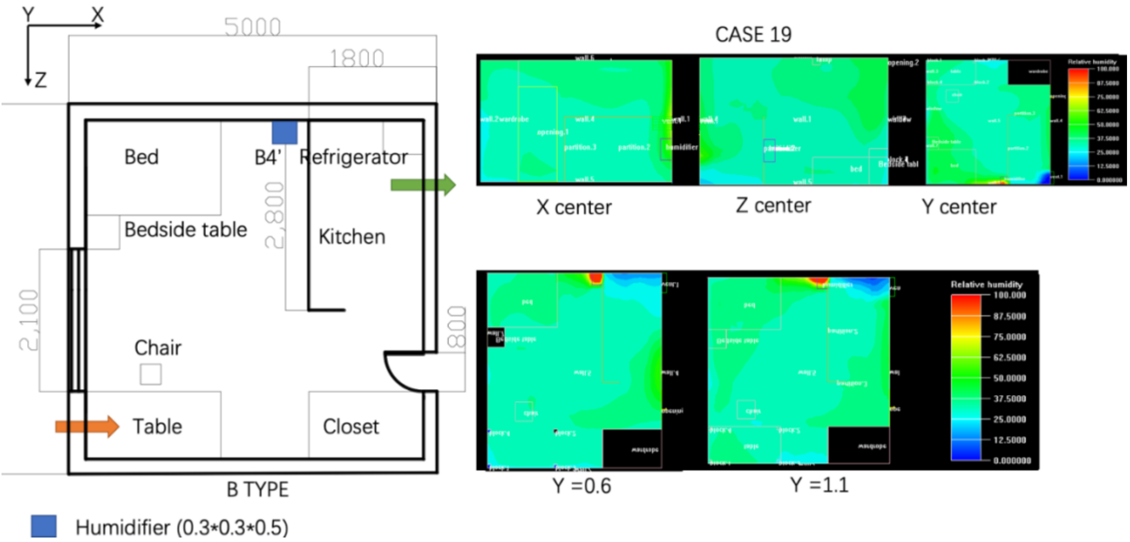

The diagram explores the effect of ventilation on the humidity distribution in a room with different air supply methods. Cases 7 and 17 show the humidifier placed in the room against the window at a height of 0.5 m above the floor.. In the case, there is significant condensation of water mist near the humidifier outlet. In terms of the overall humidity distribution in the room, the average humidity in case 7 is slightly higher than in case 17. Inevitably, the humidity on the kitchen side remained below the average range. As the humidifier is placed on the window side, in cases there is a large range of water mist condensation around the humidifier. The humidity is also at a comfortable level for the human body at a working height of 1.1m. However, the close proximity of the humidifier to the window resulted in a very visible condensation of water mist on the window side, which is very likely to have caused condensation on the window, and the area of water mist on the window side was not significantly reduced when the air supply changed. The condensation area on the window side of case 7 is slightly larger than in case 17, which means that the ventilation in case 14 mixes the room air better and to some extent inhibits the exposure phenomenon.



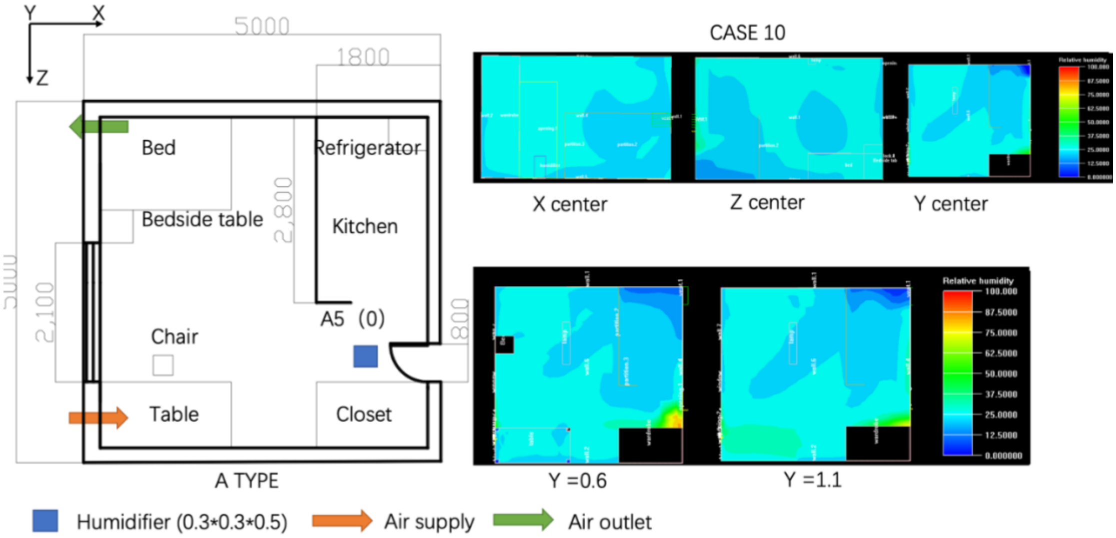



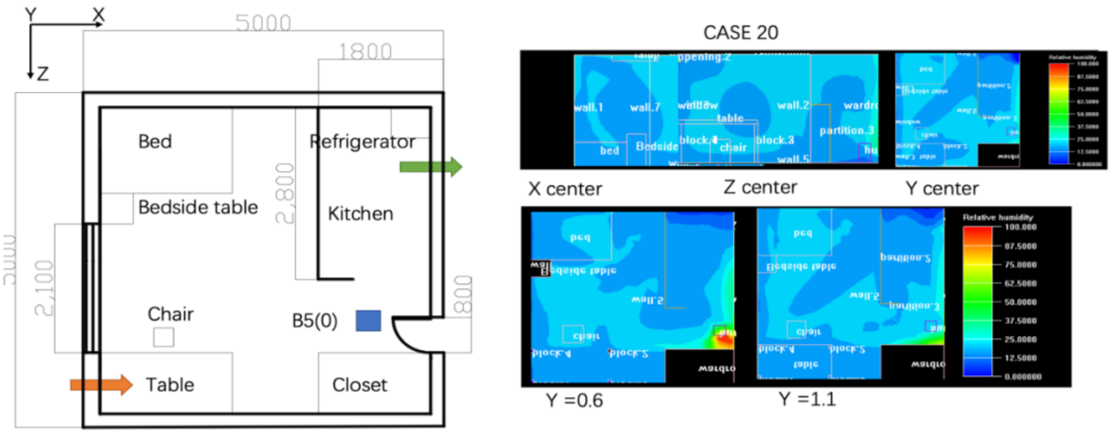

The diagram explores the effect of ventilation on the humidity distribution in a room with different air supply methods. The humidity profiles presented in both cases show that the overall humidity in the room in case 8 is not significantly different from that in case 18. However, due to the partitioning of the kitchen space from the room, the humidity level near the refrigerator on the kitchen side still did not improve. However, considering that this location is not the main place for people to stay, it is ignored for the time being. In cases 8 and 18 there was a noticeable condensation of water mist around the humidifier. As the humidifier is close to the wall, a small area of visible water mist condensation was found on the wall in case 18. The low humidity level on the outlet side was also improved. The overall average humidity in the room is within a comfortable range at a height of 1.1m at human sitting height. The condensation area on the window side of case 8 is slightly larger than in case 18, which means that the ventilation in case 14 mixes the room air better and to some extent inhibits the exposure phenomenon.



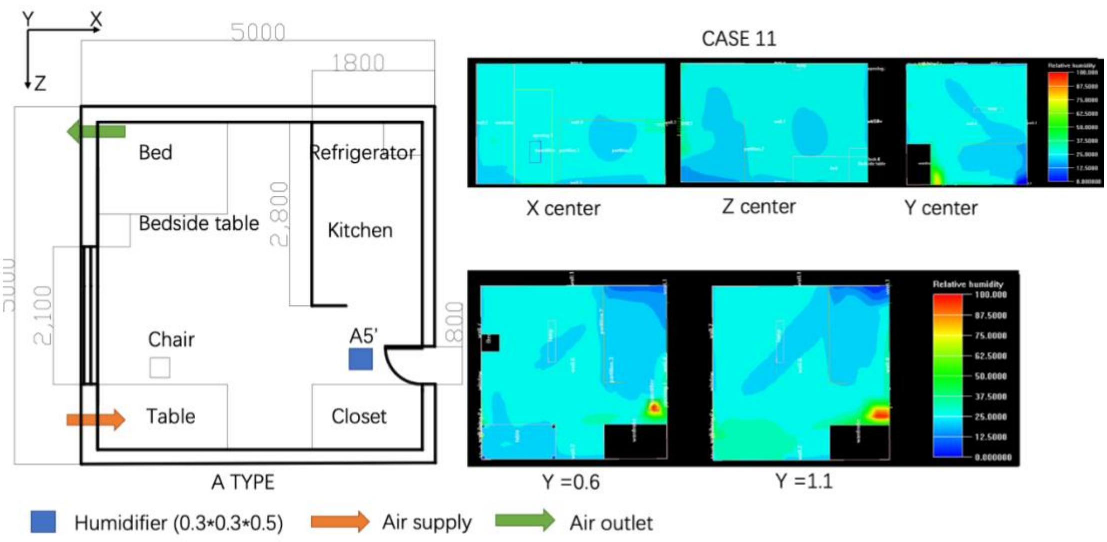



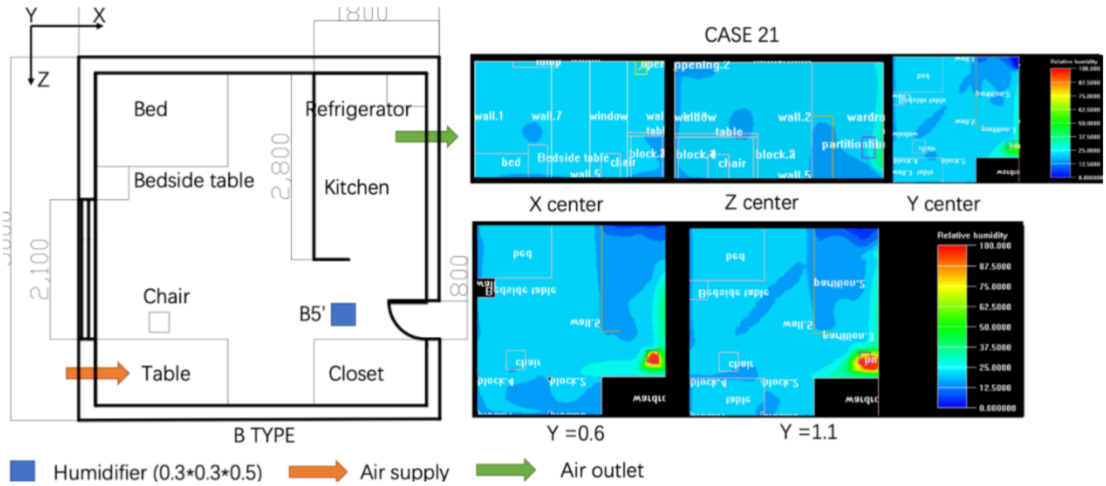

The diagram explores the effect of ventilation on the humidity distribution in a room with different air supply methods. The humidity profiles presented in both cases show that the overall humidity in the room in case 9 is significantly different from that in case 19. However, due to the partitioning of the kitchen space from the room, the humidity level near the refrigerator on the kitchen side still did not improve. However, considering that this location is not the main place for people to stay, it is ignored for the time being. In cases 9 and 19 there was a noticeable condensation of water mist around the humidifier. As the humidifier is close to the wall, a small area of visible water mist condensation was found on the wall in case 19. The low humidity level on the outlet side was also improved. The overall average humidity in the room is within a comfortable range at a height of 1.1m at human sitting height. The condensation area on the window side of case 9 is slightly larger than in case 19, which means that the ventilation in case 14 mixes the room air better and to some extent inhibits the exposure phenomenon.




Case 10 and case 20 are examples of two cases where the humidifier was placed in the doorway without ventilation. The humidity profiles show that both case 10 and case 20 do not reach the comfortable range of indoor humidity. However, the average temperature in case 10 is slightly better than in case 20. Due to the humidifier being placed by the door, the humidity distribution is not significantly different from that without the humidifier due to the infiltration air and the airflow organization of the room. In case 20, however, there was water mist condensation around the humidifier, but not in case 10. Case 10 show the humidity distribution with the same air supply as before, with the humidifier placed at the door. It is clear that the overall humidity distribution in case 10 is below the humidity range to which the human body is adapted. The placement of the humidifier does not improve the humidity in the room. The humidity in the room is low near the floor under the effect of the air conditioner, but this does not have a significant effect on the overall humidity in the room. In conclusion, when the room is supplied with air from the same side, the humidifier can be placed at a certain height around the air inlet to control the overall humidity of the room better and to prevent the occurrence of condensation on the window side to a great extent.




Case 11 and case 21are examples of two cases where the humidifier was placed in the doorway without ventilation. The humidity profiles show that both case 11 and case 21 do not reach the comfortable range of indoor humidity. However, the average temperature in case 11is slightly better than in case 21. Due to the humidifier being placed by the door, the humidity distribution is not significantly different from that without the humidifier due to the infiltration air and the airflow organization of the room. However, there was water mist condensation around the humidifier, but not in case 11. Case 21 show the humidity distribution with the same air supply as before, with the humidifier placed at the door. It is clear that the overall humidity distribution in c case 21 is below the humidity range to which the human body is adapted. The placement of the humidifier does not improve the humidity in the room. The humidity in the room is low near the floor under the effect of the air conditioner, but after raising the humidifier, we can see in case 21 that there is a clear condensation of water mist around the humidifier, but this does not have a significant effect on the overall humidity in the room.

As can be seen from Figure , when the humidifier is placed beside the door, the humidifier does not improve the indoor humidity. The humidity in ventilation mode A is higher than the humidity in ventilation mode B. And when the humidifier is placed 0.5 meters above the ground, condensation may occur on the surface of the wardrobe of the room. The reason why the indoor humidity environment is not improved can be considered in the direction of air flow. When the humidifier is placed beside the door, the air at the door of the room is not well mixed. At the same time, the air flow on the refrigerator side is also poor. However, as can be seen from Figure 5, when the humidifier is placed 0.5m from the ground, the humidity of the room is slightly higher than the humidifier placed on the ground.

## 4.4. Conclusion

This chapter establishes a mathematical model and uses FLUENT software to simulate the effects of ventilation on indoor humidity and condensation. The simulation results show that the ventilation mode and the placement of the humidifier have a great influence on the condensation distribution. The results can be summarized as follows:

1) The model can realistically simulate the effect of ventilation on the condensation distribution.

2) Within the scope of the nearby, the air just can hold limited capacity water vapor. The humidifier working efficiency was infected by the different position of humidifiers.

3) The humidity distribution in the room is related to the ventilation mode, and the flow pattern of the indoor airflow affects the distribution of indoor humidity. The room with the air supply and the return air outlets in the same direction, there is a difference in humidity on both sides of the room, and when the humidifier is used, the humidity on the other side of the room changes little. 4) The placement height of the humidifier also affects the indoor humidity distribution. When the humidifier is placed in a slightly higher position, the average humidity in the room will increase, and it will be easier to reach a comfortable humidity environment at a sleep height of 0.6 meters. But at the same time, there will be condensation of water near the humidifier.

5) The placement of the humidifier also has a great influence on the indoor humidity and humidity distribution. when the humidifier is placed beside the door, the humidifier does not improve the indoor humidity. When the humidifier is placed in the centre of the room, the humidity in the room is evenly distributed. When the humidifier is placed close to the wall, condensation occurs near the ground due to the low wall temperature.

6) From this simulation, it can be concluded that in the dry winter season, by arranging the position of the indoor vents and placing the humidifier reasonably, the humidity can reach a comfortable range while reducing the possibility of condensation.

# CHAPTER 5
# Numerical simulation analysis of office thermal environment based on Airpak

# 5. Numerical simulation analysis of office thermal environment based on Airpak

## 5.1. Introduction

There are many people who are very afraid of the cold. In winter they are most afraid of people opening the windows, and in summer they are afraid that they will catch a cold easily from the air conditioning. When an employee sits in front of an air conditioner or an open window, the temperature of some parts of his body will drop by 20 degrees. This excessive cooling can cause the employee's resistance to infection to drop and cause a red light to go on for his or her health. Such a situation is most likely to cause colds. The sweltering heat in the office is a direct result of the rapid decline in productivity. Two-thirds of those interviewed admitted that people are unable to maintain a positive work ethic when the office gets hot. Moreover, the carbon dioxide exhaled by employees in a hot and stuffy office can quickly build up. Numerous studies have shown that the increase in carbon dioxide levels leads to reduced brain function and even mental fatigue. A survey published by Harvard University proves that a good fresh air system can effectively solve the problem of dullness

HEADHUNTER, a leading European executive search firm, once held a large-scale white-collar survey interview on the topic "Are you satisfied with your office environment?" A total of 3,000 office workers took part in the interview, the results of which were as follows.

HeadHunter interview findings:

1. 90% of offices are equipped with cooling and heating equipment, and even humidifiers. But this does not ensure a good office environment.
2. Although the offices are fully air-conditioned, 68% of the people interviewed said that they often felt a bit depressed and dull in the office, and most of them even revealed that 66% of the people in the office often had arguments because of the office environment.
3. 46% of the office personnel interviewed said they would fight for the right to set the air conditioning temperature. And, one in ten believe that air conditioning is a direct danger to their health. This belief actually makes sense: 50% admit that cold air conditioning is usually the culprit for colds.
4. For 17 percent, the most critical issue is whether the office windows should be opened. The most common complaints from office workers were: too hot, too cold and too dull. Of course, in the summer more people complained that the office was unusually hot, while in the winter more people complained about the chilly draught inside. However, due to the prevalence of air conditioning in offices, 17% of people still complained that it was particularly cold in the summer, while 18% complained that it was hot in the winter due to the heating system in the winter.

The results of these interviews make us realise that neither turning on the air conditioning nor opening the windows creates a comfortable office environment for employees.

This statistic makes us realise that turning on the air conditioning or creating ventilation does not create a comfortable office environment.

## 5.2. The summary of CFD simulation

### 5.2.1. Establishment of a basic physical model

In this chapter the office is selected for simulation. The office is 9m long and 8m wide. The external walls are all glass curtain walls and the room has two doors, 2100mm*1500mm. in the all-air primary return air system (PLAN A), there are four air supply outlets 0.3m*0.3m, the air supply speed is 2.7m/s, the air supply temperature is 29°C, and there are two return air outlets 0.6m*0.3m. in the VRV multi-connector plus fresh air air-conditioning system (PLAN B), the central part of the indoor unit is 0.6*0.6 0.6m return air outlet, set for natural return air, surrounded by 0.5*0.5m air supply outlet, air supply direction and horizontal plane angle of 45 degrees, outlet air speed 4.5m/s, air supply temperature 29°C,, new air outlet 0.18*0.18m, air speed 2.0m/s, air supply temperature 20 degrees Celsius. The physical model is shown in Figure 5-1. The office area per person in the high-end office is 8m2/person, so each office is allocated to 8 office staff and 8 computers. To ensure lighting, the office is equipped with 8 electric lamps. The indoor and outdoor design parameters are shown in Table 5-1.

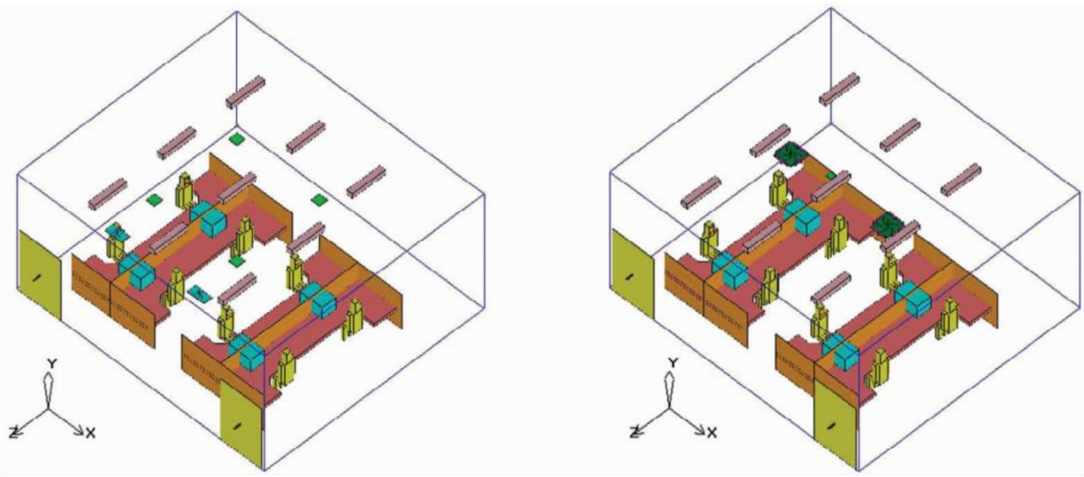


Figure 5-1 office physical model

Table 5-1 Air conditioning setting parameters

| **Air conditioning outdoor weather parameters** | | | |
|---|---|---|---|
| **Calculated outdoor temperature in winter** | **Calculated daily relative humidity outdoors in winter** | | |
| **5 °C,** | 60% | | |
| **Air conditioning interior design parameters** | | | |
| **SEASON** | **TEMP.（°C,）** | **RH%** | **Average velocity of air flow（m/s）** |
| **Winter** | 29 | 50 | ≤0.2 |

Assuming that the indoor air is an incompressible fluid, the indoor heat source only needs to be considered in terms of people, computers and lamps to dissipate heat.The PMV-PPD calculation is based on the human body sitting still, with a human metabolic rate of 58w/m2. The system boundary conditions are shown in Table 5-2.

Table 5-2 Model Boundary Conditions.

| **Boundary** | **Parameter Setting** |
|---|---|
| Room | 9*8*4, outdoor temperature -2°C |
| Wall | Dehumidification, overall heat transfer coefficient 1.86W/m2・K. Outdoor temperature: 5°C |
| Initial environment | Temp: 22 °C, RH: 60% |
| Ceiling and Floor | Insulation |
| Computer | 8,0.4*0.4*0.4 110W |
| People | 8, 1.73*0.3*0.2,75W |
| Light | 8,1.2*0.2*0.2,35W |

### 5.2.2. Simplification of the physical model

If the simulated state following the actual state of lab, the error between the actual measurement and the simulation experiment will be minimized. However, due to the accuracy of the model will affect the division of the mesh, the operation of the computer's performance requirements is high, It is difficult for ordinary computers to meet the requirements, so in order to simulate the smooth progress of this model I affect the room airflow organization smaller objects to a certain degree of simplification, thereby reducing the operation of the computer performance requirements. 5 points simplifying assumption were made for the model：

1.The effects of gravity fields on air and water vapors are ignored;

2.Both air and water vapor are considered to be incompressible fluids with a constant density;

3.When the water vapor evaporates from the water surface, only latent heat exchange is performed, and sensible heat exchange is not considered;

4.When water vapor condenses on the wall, the heat release ignored from the phase change process;

5.Wall surfaces moisture absorption capacity is not considered during wall condensation.

## 5.3. Mathematical method

In the building indoor environment, generally the water vapor in the humid air is in a superheated state and the content is very low, so at a certain extent, it can be approximated as the ideal gas. At the same time the density is small, the diffusion of wet air can be ignored.

The basic laws of conservation of heat and mass transfer between the fluid and the fluid and between the fluid and the solid interface are: mass conservation law, momentum conservation law, and energy conservation law. The mathematical models that generally describe these conservation laws are the corresponding governing equations, namely the mass continuity equation, the momentum equation, and the energy equation. As follows：

Mass continuity equation：

$$\frac{\partial \rho}{\partial_t} + \nabla \cdot (\rho U) = 0 \tag{5-1}$$

In the formula, ρ is the fluid density, kg/$m^3$； U is the gas velocity vector, $U_{x,y,z}$.

In this study, considering air as an incompressible fluid, the continuity equation can be simplified to:

$$\nabla U = 0 \tag{5-2}$$

Momentum equation:

$$\frac{\partial(\rho U)}{\partial_t} + \nabla \cdot (\rho U \otimes \mathrm{U}) = -\nabla P + \nabla \cdot \tau + \rho_g \tag{5-3}$$

where ⊗ is the outer product: $U \otimes \mathrm{V} = \mathrm{U}V^T$,$\rho$ is the fluid density ,U is the flow velocity vector,

$\nabla$ ·is the divergence ,P is the pressure,t is the time$\tau$ is the derivatoric stress tensor, which has order two,$\tau = \mu(\nabla U + (\nabla U)^T) - \frac{2}{3}\delta\nabla \cdot U$

g represents body accelerations acting on the continuum, for example gravity, inertial accelerations, electrostatic accelerations.

The energy equation：

$$\frac{\partial \rho h_{tot}}{\partial t} - \frac{\partial_p}{\partial t} + \nabla \cdot (\rho U h_{tot}) = \nabla \cdot (\lambda \nabla T) + \nabla \cdot (U \cdot \tau) + U \cdot S_M + S_E \tag{5-4}$$

Where：

$h_{tot}$ is the total enthalpy of the fluid, $h_{tot} = h + 1/2U^2$ J/kg.

$h$ is the static enthalpy of the fluid, J/kg. And $\lambda$ is the thermal conductivity of the fluid, $W/(m \cdot K)$.

$S_E$ is the source of fluid energy.

There is no chemical reaction in the numerical simulation process of this subject, and the transport equation of the unreacted material is as shown in the following formula:

$$\frac{\partial(\rho Y_i)}{\partial_t} + \nabla \cdot (\rho \upsilon Y_i) = -\nabla J_i \tag{5-5}$$

$Y_i$ is the mass fraction of each substance predicted by FLUENT by the convective diffusion equation of the i^th substance; $J_i$ is the diffusion flux of substance i, which is produced by the concentration gradient?

When the moving fluid is laminar:

$$J_i = -\rho D_{i,m} \nabla Y_i \tag{5-6}$$

Where: $D_{i,m}$is the diffusion coefficient of the first species in the mixture.

When the moving fluid is turbulent:

$$J_i = -\left\{\rho D_{i,m} + \frac{\mu_t}{Sc_t}\right\} \nabla Y_i \tag{5-7}$$

Where:$Sc_t$ is the turbulent Schmidt number

In the formula “d” is the absolute humidity. The absolute humidity between the mass of water vapor present in moist air to the mass of dry air. Absolute humidity is normally expressed in kilograms of water vapor per kilogram of dry air.

Absolute humidity expressed by mass:

$$\mathrm{d} = \frac{\mathrm{m_w}}{\mathrm{m_a}} \tag{5-8}$$

Where: d = absolute humidity; $\mathrm{m_w}$= mass of water vapor (kg):

$\mathrm{m_a}$= mass of dry air (kg)

The FLUENT 14.0 turbulence module (Standard k-ε) is used to analyze the flow field characteristics. The component transport module (Species Transport) and the multiphase flow module (Multiphase) simulated the mass transfer phase transition of water vapor in the air. The governing equation is as follows:

$$\frac{\partial(\rho\emptyset)}{\partial_\tau} + div(\rho u\emptyset) = div(\Gamma grad\phi) + s \tag{5-9}$$

Where: $\phi$is a general variable;vΓ is a generalized diffusion coefficient; s is a generalized source term.

Wall condensation conditions: When the absolute humidity $D_{air}$ of the air node around the wall surface is higher than the saturated absolute humidity $Dwall$ corresponding to the wall temperature, the wall surface is considered to be condensation.

Conditions for no condensation on the wall: Any time τ, when the absolute humidity $Dair$ of the air node around the wall is less than the saturated absolute humidity $Dwall$ corresponding to the wall temperature, the wall is not considered to be condensation

$$Dwall < Dair \tag{5-10}$$

### 5.3.1. Distribution hypothesis and convergence principle of dependent variable

1. Distribution hypothesis of dependent variable

This topic adopts different distribution hypotheses for different dependent variables. When the grid is sufficiently fine, the variation of the dependent variable between the grids decreases, and the impact of such different assumptions will also be reduced. According to different calculation actual situations, the first-order or second-order welcome style is adopted to ensure the accuracy and stability of the calculation results.

2. The principle of convergence

The convergence criterion for the energy equation is generally taken as: $1 \times 10^{-6}$ (for example: the convergence criterion for temperature); the convergence criterion for the flow equation is generally taken as: $1 \times 10^{-3}$ (for example: the convergence criterion for velocity). The convergence criterion also reflects the accuracy of the simulation calculation.

3. Determination of relaxation factor

In the process of calculating by successive iteration method, the choice of relaxation factor is a more critical link. A properly selected relaxation factor can accelerate convergence. Improper selection may cause calculation oscillation or divergence. As far as the solution of the entire nonlinear problem is concerned, there is no complete theory to judge whether the iterative solution method can obtain a convergent solution. The sub-relaxation iteration method is used in FLUENT software to accelerate convergence. Many scholars have studied the determination of the optimal relaxation factor, but they all aim to directly find the optimal relaxation factor. The selection of such relaxation factors is quite complicated. The common method in engineering is to choose different relaxation factors for trial calculation according to the convergence speed of the iteration process, constantly modify, and gradually find the best, until it is satisfied, then fix it and continue the iteration, in order to achieve the purpose of accelerating convergence.

In the calculations in this chapter, the pressure relaxation factor is between 0.1 and 0.3, the momentum relaxation factor is between 0.1 and 0.5, the temperature and viscosity relaxation factor is between 0.7 and 1.0, and the mass force relaxation factor is 0.1. The relaxation factor of the turbulent dissipation rate is 0.3 to 0.5. Based on the microscopic analysis of the characteristics of air turbulence, the main method is to adopt the indoor zero equation model and use the controlled volume dispersion method to discrete the control equations. The professional software Airpak2.1 introduced by Fluent is used for simulation calculation. Airpak2.1 uses FLUENT5.6.6 CFD solver for calculation.

### 5.3.2. Simplification of the physical model

1.The effects of gravity fields on air and water vapors are ignored;

2.Both air and water vapor are considered to be incompressible fluids with a constant density;

3.When the water vapor evaporates from the water surface, only latent heat exchange is performed, and sensible heat exchange is not considered;

4.When water vapor condenses on the wall, the heat release ignored from the phase change process;

5.Wall surfaces moisture absorption capacity is not considered during wall condensation.

## 5.4. Post-processing and analysis of simulation results

### 5.4.1. Velocity field analysis (air flow analysis)

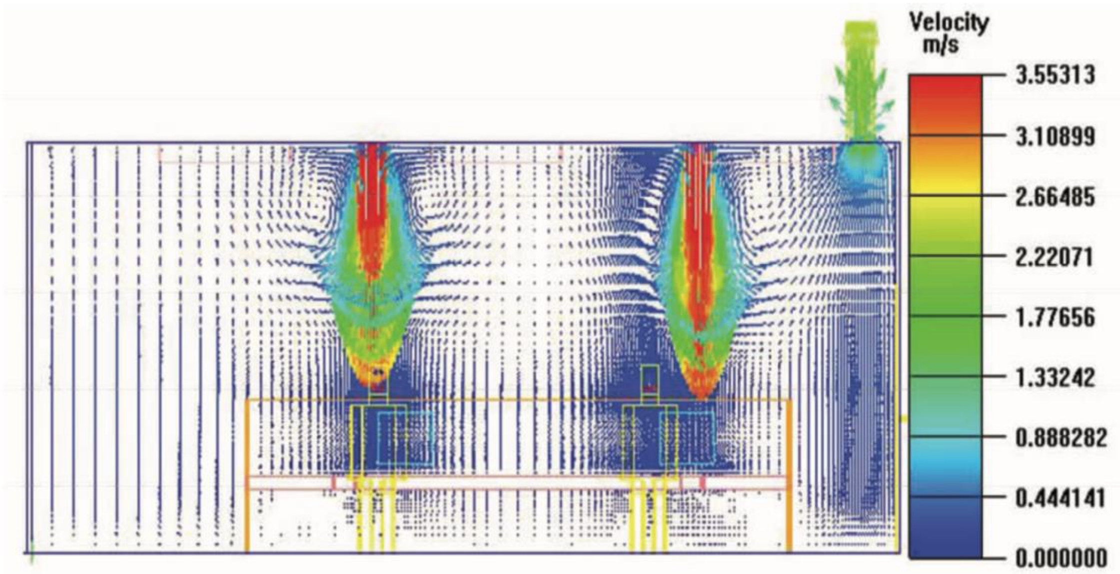



Figure 5-2 Velocity vector at X=3m plane of case A

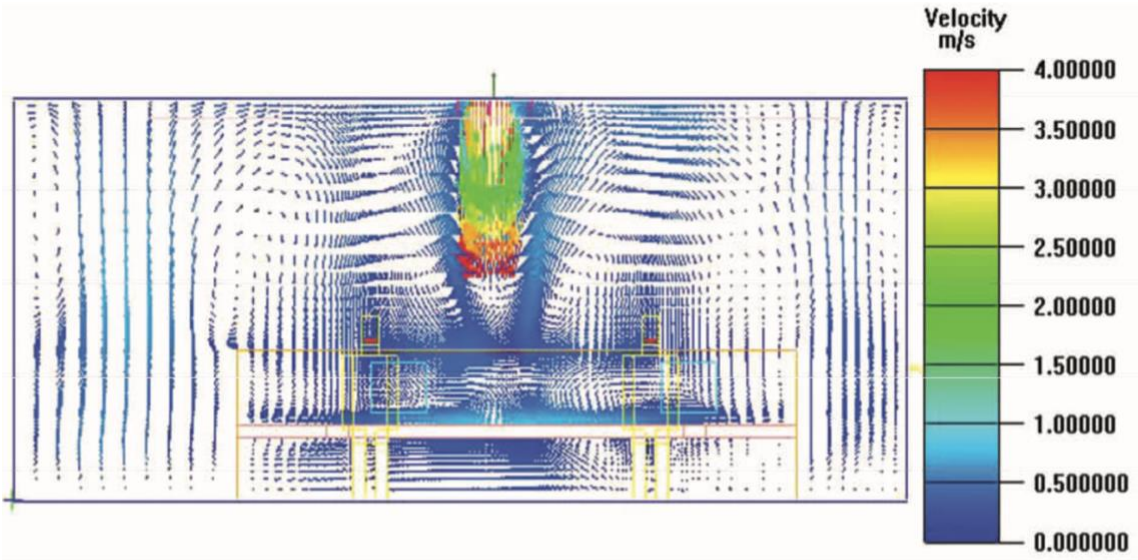



Figure 5-3 Velocity vector at X=3m plane of case B

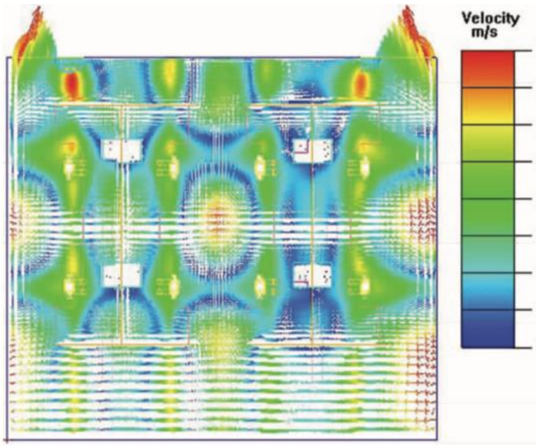



Figure 5-4 Velocity vector at Y=1m plane of case A

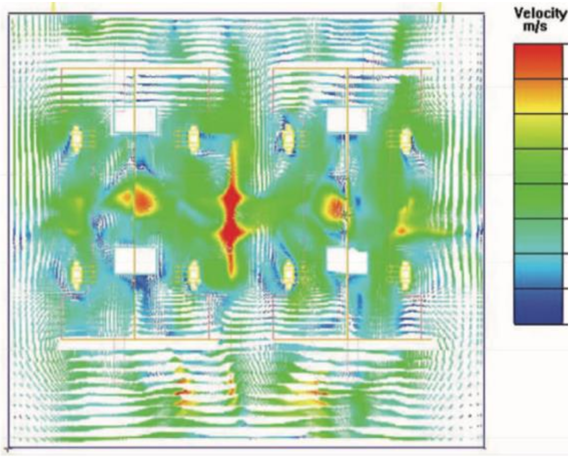



Figure 5-5 Velocity vector at Y=1m plane of case B

As can be seen from the diagram, the airflow under the action of the two options A and B reaches the working area for mixing and heat exchange, then due to the obstacle of the computer desk and other obstacles, the computer, human body, heat source drive up through the return air outlet discharge.

Case A indoor arrangement of four air outlets and two return air outlets, indoor vortex less, human breathing area wind speed of about 0.05 ~ 0.25m/s, personnel feel comfortable, local wind speed reached 0.45m/s; B program indoor arrangement of two indoor units and a new air outlet, air supply is not as uniform as A program, VRV indoor unit and the joint action of fresh air, making the indoor air disturbance is more violent, vortex relatively more The wind speed in the human breathing zone is about 0.08~0.35m/s. Individuals have a slight sense of blowing wind and the local wind speed reaches 0.7m/s. People with high wind speed should avoid staying here for a long time. In terms of air homogeneity, option a has a uniform air supply with insignificant room disturbance. Whereas, due to the combined effect of the VRV air conditioning system and the fresh air sipply, the airflow disturbance of option b is noticeable.

### 5.4.2. Temperature field analysis

In an air-conditioned room, the temperature difference between 1.1m and 0.1m above the floor in the work area should not be greater than 3°C within a comfortable range (in the case of sitting work). The velocity fields at y=0.1m (ankle height) and y=1.1m (personnel sitting height) were selected for comparative analysis under the two scenarios.

The temperature of the nine points uniformly distributed in the working area (from top to bottom and from left to right) in Figures 5-6.5-7 are summarised in Figure 5-8. The analysis shows that the average temperature of the y=1.1m section is 23.7°C and the average temperature of the y=0.1m section is 23.6°C. The maximum temperature difference between 1.1m and 0.1m above the ground in the working area is 0.4°C, so the comfort of the personnel in the working area is stronger.

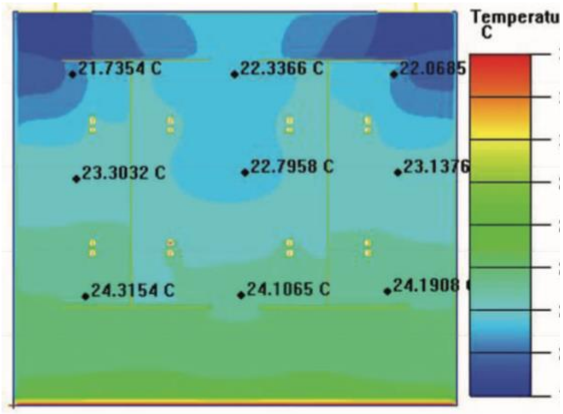



Figure 5-6 Temperature contours at Y=0.1m plane of Case A

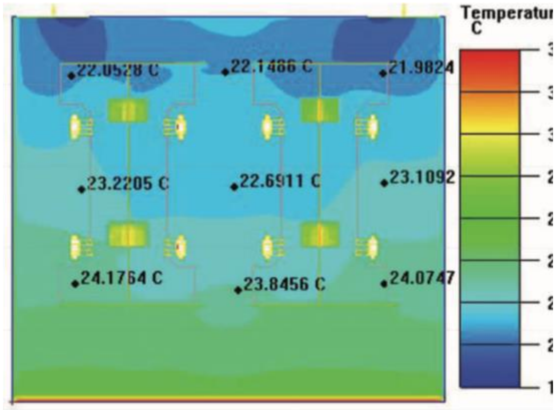



Figure 5-7 Temperature contours at Y=1.1m plane of Case A

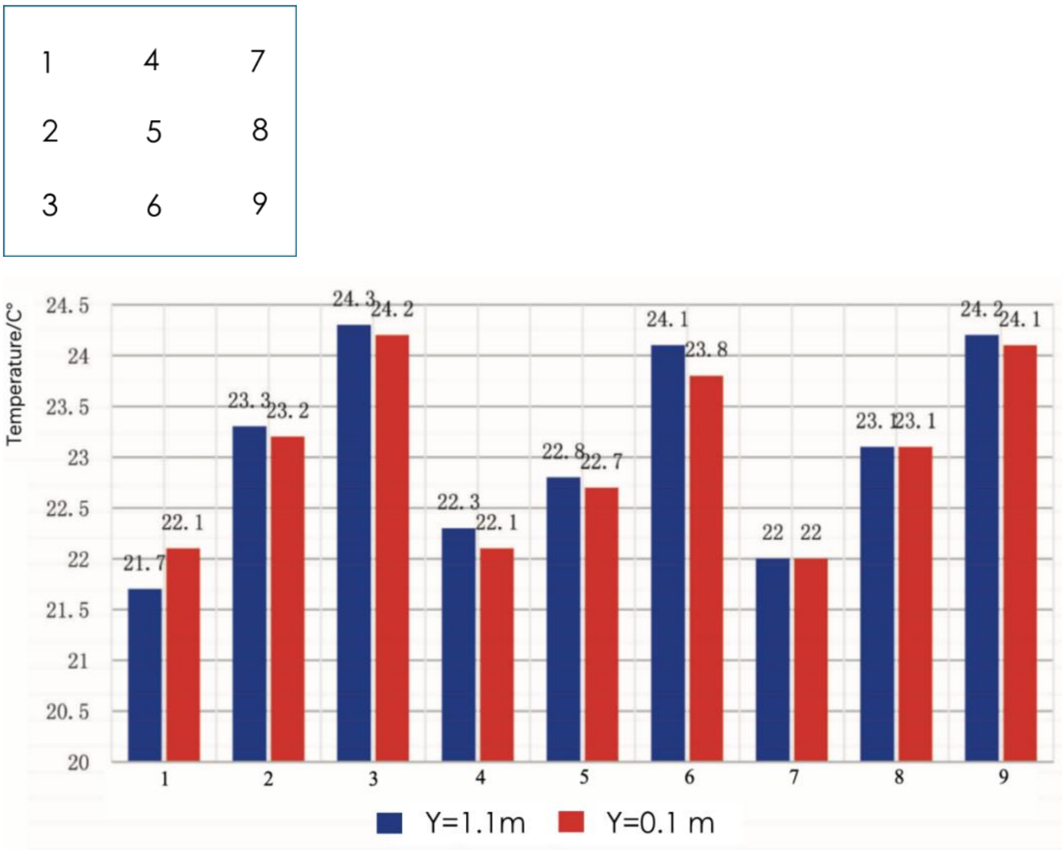



Figure 5-8 The height plane of temperature contours of Case A

Figure 5-9.5-10 in the work area uniformly distributed in the nine points (in turn from top to bottom, from left to right) temperature summary in Figure 5-11, analysis can be seen, y = 1.1m section of the average temperature of 25.1 ℃, y = 0.1m section of the average temperature of 24.4 ℃, the work area above the ground 1.1m and 0.1m between the maximum temperature difference of 1.4 ℃, so the work area personnel comfortable. From the diagram we can clearly see that the temperature difference is greater for Option B than for Option A. In terms of comfort, although both are in the comfort range, the option offers a better airflow organization.

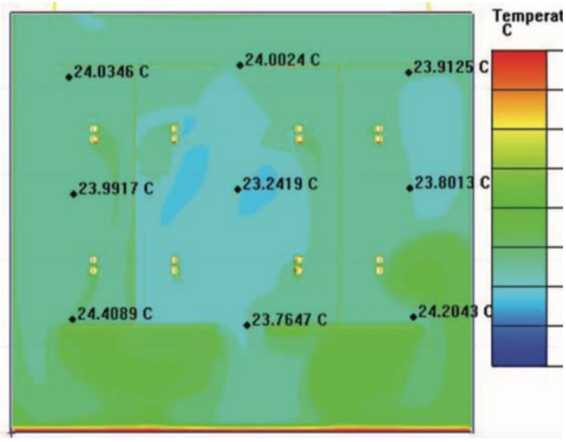


Figure 5-9 Temperature contours at Y=0.1m plane of Case B

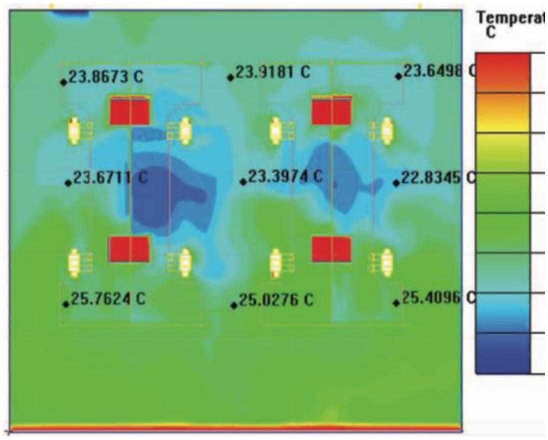


Figure 5-10 Temperature contours at Y=1.1m plane of Case B

| | | |
|---|---|---|
| 1 | 4 | 7 |
| 2 | 5 | 8 |
| 3 | 6 | 9 |

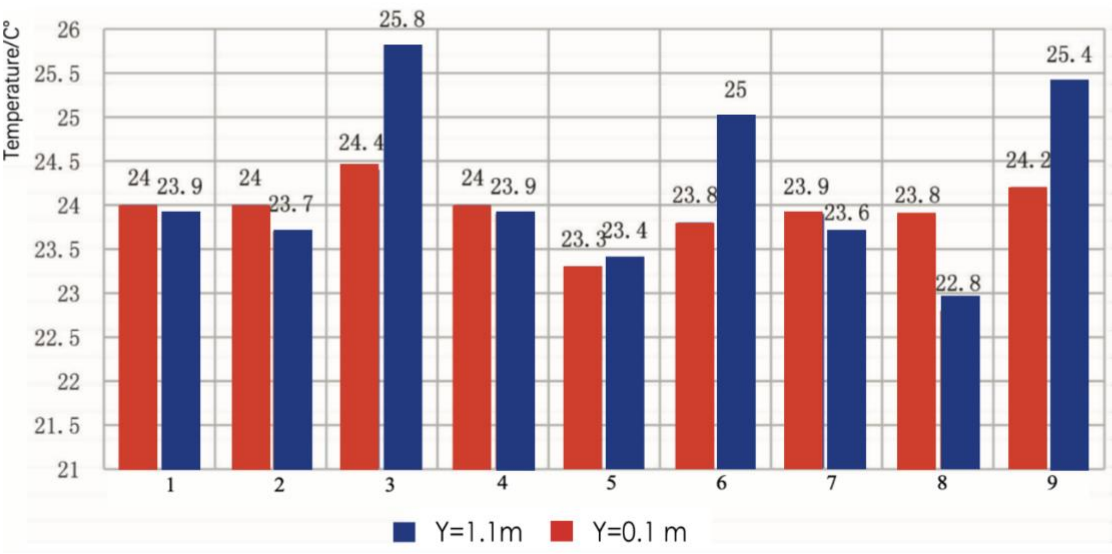



Figure 5-11 The height plane of temperature contours of Case B

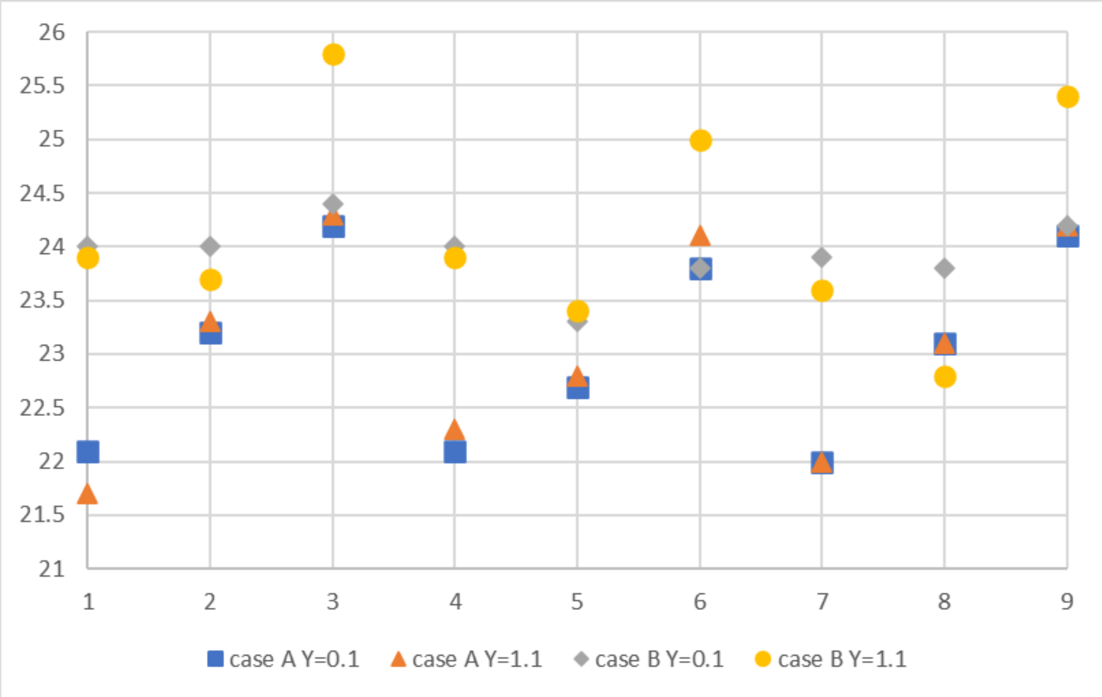



Figure 5-12 The height plane temperature contours of case A and case B

To better understand the differences in room temperatures between case A and case B, and the reasons for the temperature differences that exist, the temperatures in scenarios a and b are integrated in 5-12.

From the analysis, it is clear that the temperature difference between the two scenarios is not significant at y=0.1m, and at y=1.1m the temperature of scenario b is significantly higher, although the room temperature shows a low temperature near the ground from time to time, but the comparison of the two scenarios is, we guess, due to the larger area of the room and the more violent disturbance of the indoor air, this point is also reflected in the velocity distribution diagram. In option b the airflow organisation is more fluid, with more vortices, and there are individual locations with higher air velocities and uneven airflow. The airflow is not as efficient as it could be in the room, and the airflow does not take away the heat from the equipment and the human body in time, making the temperature at the height of the human body in a sitting position slightly higher. In this regard, the temperature difference in option b is larger.

### 5.4.3. Humidity analysis

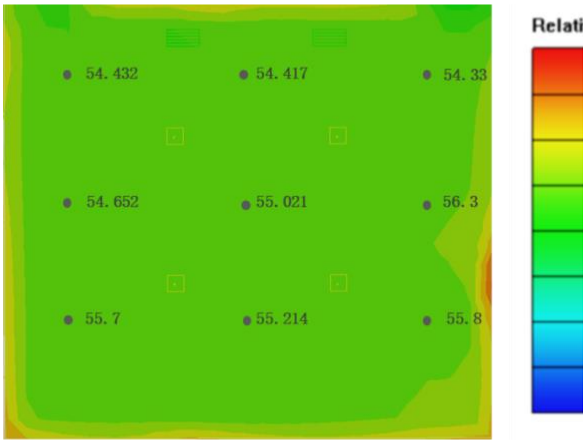



Figure 5-13 Relative humidity of case A

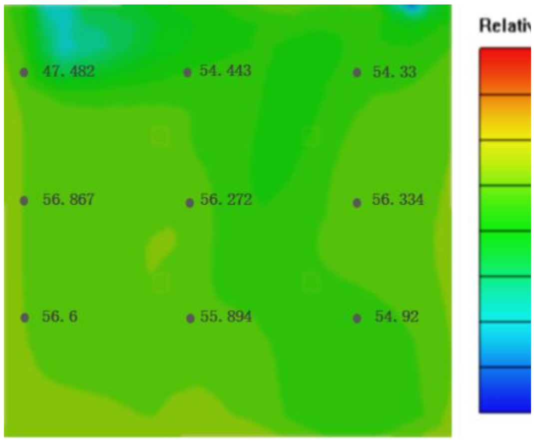
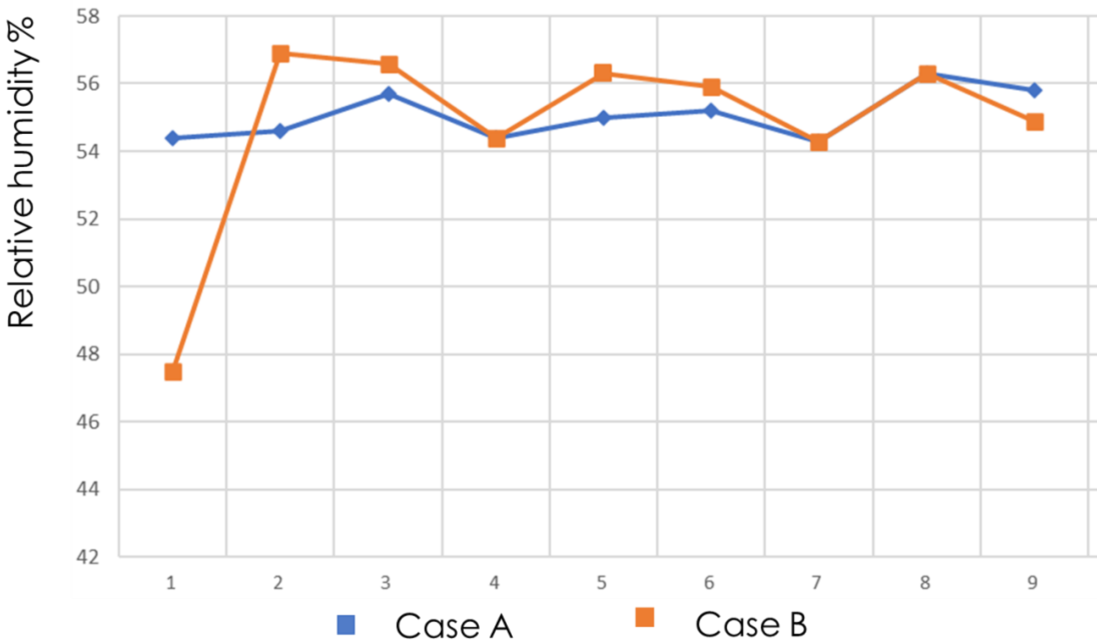



Figure 5-14 Relative humidity of case B




As can be seen in the diagram, both Case A and Case B provide a good indoor humidity environment. The indoor humidity range at a height of 1.1m is between 40% and 60%. Case B has a higher average humidity than Case A, except at point 1. The relative humidity for case B are much higher than those for case A. This is another way of verifying that case B is deficient in terms of airflow uniformity and air mixing.And due to excessive airflow disturbances and vortexes, in Case B the two corners of the room away from the door have a lot of stagnant air, which in today's epidemic environment does not allow the room air to exit the room quickly enough. There is a serious shortage of fresh air in the room. For the people in the room, the air in the room can feel dull to a large extent. This affects productivity. The fresh air sent through the air supply vents is also not well distributed in the rooms. The efficiency of the room ventilation is low.

## 5.4.4. Air mean age analysis

Age of air is the age of air mass (Age of air), refers to the air mass since the entry into the room to a point in the room experienced time, reflecting the freshness of indoor air, it can be a comprehensive measure of the room ventilation effect, is an important indicator to evaluate the quality of indoor air. The concept of air age was first introduced by Sandberg in the 1980s. By definition, air age refers to the time when air enters a room. When pollutants are evenly distributed in a room and the air supply is fully fresh, the smaller the air age at a point, the fresher the air at that point and the better the air quality. It also reflects the room's ability to remove pollutants, with a room with a small average air age being more capable of removing pollutants. Due to the obvious physical significance of air age, it is widely used as an important indicator of the freshness of air in air-conditioned rooms and the ability to change air. Traditionally the air age concept has only considered the interior of the room, the air age at the room inlet is considered to be 0 (100% fresh air). In order to consider the effect of the entire ventilation system including return air, mixed air and flow in the ducts, Tsinghua University has proposed the concept of full air age. The time that the air micro-clusters experience since entering the ventilation system; the air age at the entrance of the room is taken as 0 and the resulting air age is called the room air age. Compared to the room air age, the full air age can be seen as an absolute parameter and the full air ages of different rooms can be compared.

As air age affects more the height at which a person sits, we have selected only 9 points at a height of 1.1m for the air age analysis. The air age values for case B are much higher than those for case A. This is another way of verifying that case B is deficient in terms of airflow uniformity and air mixing.

And due to excessive airflow disturbances and vortexes, in Case B the two corners of the room away from the door have a lot of stagnant air, which in today's epidemic environment does not allow the room air to exit the room quickly enough. There is a serious shortage of fresh air in the room. For the people in the room, the air in the room can feel dull to a large extent. This affects productivity. The fresh air sent through the air supply vents is also not well distributed in the rooms. The efficiency of the room ventilation is low.

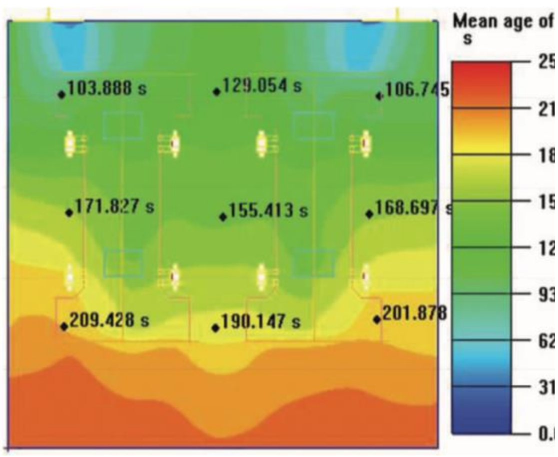



Figure 5-15 Air mean age at Y=1.1m plane of case A

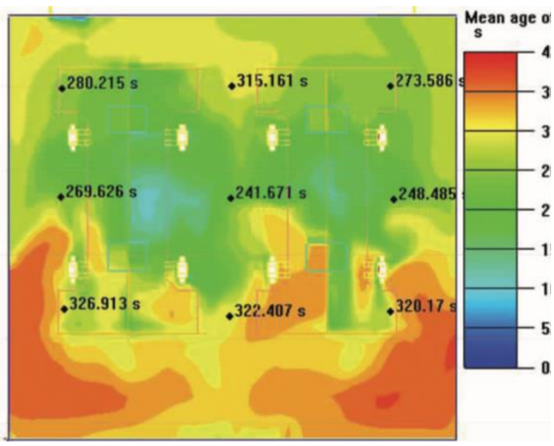



Figure 5-16 Air mean age at Y=1.1m plane of case B

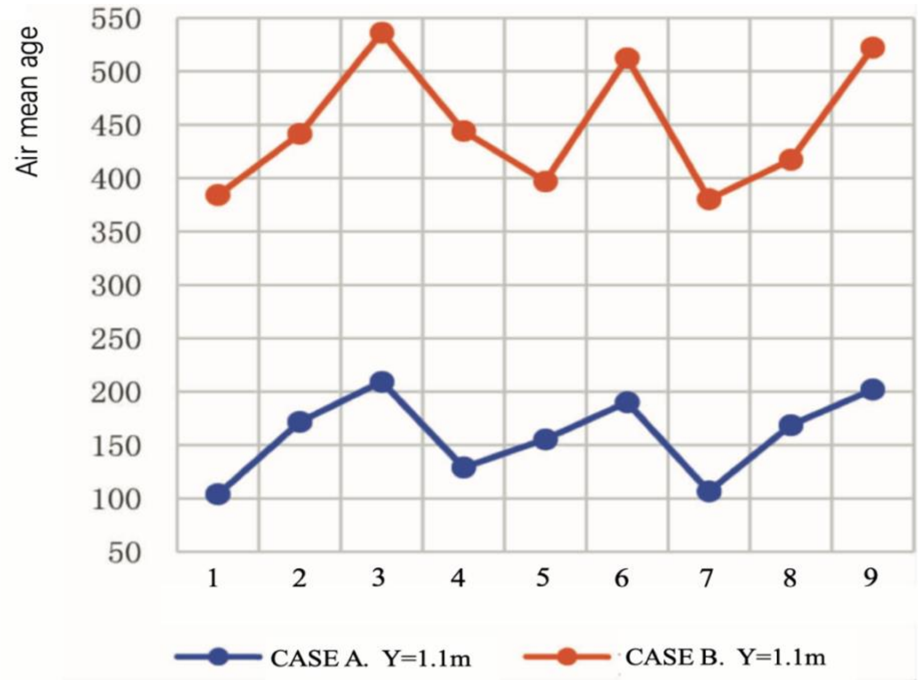


Figure 5-17 Air mean age at Y=1.1m plane of case A and case B

The air age values for the nine points evenly distributed in Figures 5-13, 5-14 are summarised in Figure 5-15. The air age values in scheme a are 104-210s, with an average value of 166s, while in scheme b the air age values are 240s-327s, with an average value of 299s. During the flow of air from the air supply into the room, indoor pollutants are continuously mixed and the cleanliness of the air decreases, but due to the smooth airflow organisation in scheme a, the air exchange efficiency is better. In scheme b, however, the airflow disturbance area is relatively large, the replacement effect is slightly worse and the air quality is slightly poorer.

## 5.4.5. PMV-PPD analysis

To better understand the model that produces values for the predicted mean vote (PMV) and the predicted percentage of dissatisfied (PPD), we should start at the beginning, with the man who created them, Povl Ole Fanger. He hypothesized that human thermal comfort was based on one's skin temperature and their sweat secretion, and that one could only be considered 'comfortable' if these two factors were balanced within a narrow range of acceptability.

As humans, our thermoregulatory system modifies our temperature through involuntary responses. For example, by sweating in high temperatures or shivering in cold temperatures to keep us thermally balanced and to avoid local discomfort. The human body can adapt to the external environment up to a certain range, but as soon as the limits are reached, the body's responses are perceived as uncomfortable. Through climate chamber experiments, Fanger's theory evolved to declare that thermal comfort could be found from evaluating the metabolic rate, clothing insulation and environmental conditions of an individual.

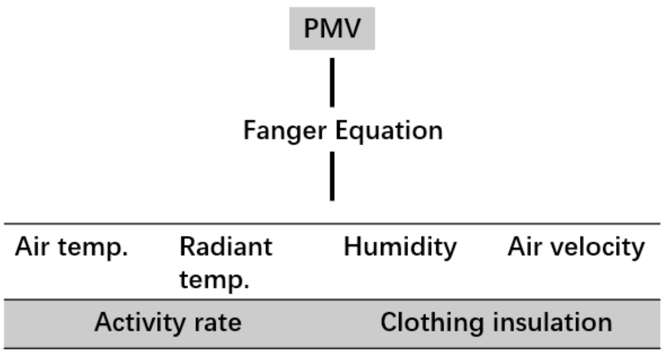



Figure 5-18 Environmental and personal factors that influence thermal comfort

Thermal comfort is defined as "that condition of mind that expresses satisfaction with the thermal environment" in the globally recognized ASHRAE 55 and ISO 7730 standards for evaluating indoor environments. To assess this condition, engineers must first determine the thermal sensation or thermal balance inhabitants of an indoor environment may feel in tangent with the thermal dissatisfaction experienced by occupants. These comfort limits can be expressed by the PMV and the PPD indices.

PMV is an index that aims to predict the mean value of votes of a group of occupants on a seven-point thermal sensation scale. Thermal equilibrium is obtained when an occupant's internal heat production is the same as its heat loss. The heat balance of an individual can be influenced by levels of physical activity, clothing insulation, as well as the parameters of the thermal environment. For example, thermal sensation is generally perceived as better when occupants of a space have control over indoor temperature (i.e., natural ventilation through opening or closing windows), as it helps to alleviate high occupant thermal expectations on a mechanical ventilation system.

Within the PMV index, +3 translates as too hot, while -3 translates as too cold, as depicted below.

| PMV Value | Comfort Condition |
|---|---|
| -3 | Cold |
| -2 | Cool |
| -1 | Slight Cool |
| 0 | Neutral |
| 1 | Slight Warm |
| 2 | Warm |
| 3 | Hot |

Figure 5-19 PMV thermal sensation ruler

Different methods outlined in the ASHRAE 55 and ISO standards for certain types of environments can be used to assess and gather information for various combinations of metabolic rate, insulation, temperature, airspeed, mean radiant temperature, and relative humidity that factor into PMV.

$$PMV = \left[0{,}303 \cdot \exp(-0{,}036 \cdot M) + 0{,}028\right] \cdot \left\{ \begin{array}{l} (M-W) - 3{,}05 \cdot 10^{-3} \cdot \left[5\,733 - 6{,}99 \cdot (M-W) - p_a\right] - 0{,}42 \cdot \left[(M-W) - 58{,}15\right] \\ -1{,}7 \cdot 10^{-5} \cdot M \cdot (5\,867 - p_a) - 0{,}001\,4 \cdot M \cdot (34 - t_a) \\ -3{,}96 \cdot 10^{-8} \cdot f_{cl} \cdot \left[(t_{cl}+273)^4 - (\bar{t}_r + 273)^4\right] - f_{cl} \cdot h_c \cdot (t_{cl} - t_a) \end{array} \right\} \quad (1)$$

$$t_{cl} = 35{,}7 - 0{,}028 \cdot (M-W) - I_{cl} \cdot \left\{3{,}96 \cdot 10^{-8} \cdot f_{cl} \cdot \left[(t_{cl}+273)^4 - (\bar{t}_r + 273)^4\right] + f_{cl} \cdot h_c \cdot (t_{cl} - t_a)\right\} \quad (2)$$

$$h_c = \begin{cases} 2{,}38 \cdot \left|t_{cl} - t_a\right|^{0{,}25} & \text{for} \quad 2{,}38 \cdot \left|t_{cl} - t_a\right|^{0{,}25} > 12{,}1 \cdot \sqrt{v_{ar}} \\ 12{,}1 \cdot \sqrt{v_{ar}} & \text{for} \quad 2{,}38 \cdot \left|t_{cl} - t_a\right|^{0{,}25} < 12{,}1 \cdot \sqrt{v_{ar}} \end{cases} \quad (3)$$

$$f_{cl} = \begin{cases} 1{,}00 + 1{,}290\, I_{cl} & \text{for } I_{cl} \leqslant 0{,}078 \text{ m}^2 \cdot \text{K/W} \\ 1{,}05 + 0{,}645\, I_{cl} & \text{for } I_{cl} > 0{,}078 \text{ m}^2 \cdot \text{K/W} \end{cases} \quad (4)$$

where

$M$ is the metabolic rate, in watts per square metre (W/m²);

$W$ is the effective mechanical power, in watts per square metre (W/m²);

$I_{cl}$ is the clothing insulation, in square metres kelvin per watt (m² · K/W);

$f_{cl}$ is the clothing surface area factor;

$t_a$ is the air temperature, in degrees Celsius (°C);

$\bar{t}_r$ is the mean radiant temperature, in degrees Celsius (°C);

$v_{ar}$ is the relative air velocity, in metres per second (m/s);

$p_a$ is the water vapour partial pressure, in pascals (Pa);

$h_c$ is the convective heat transfer coefficient, in watts per square metre kelvin [W/(m² · K)];

$t_{cl}$ is the clothing surface temperature, in degrees Celsius (°C).

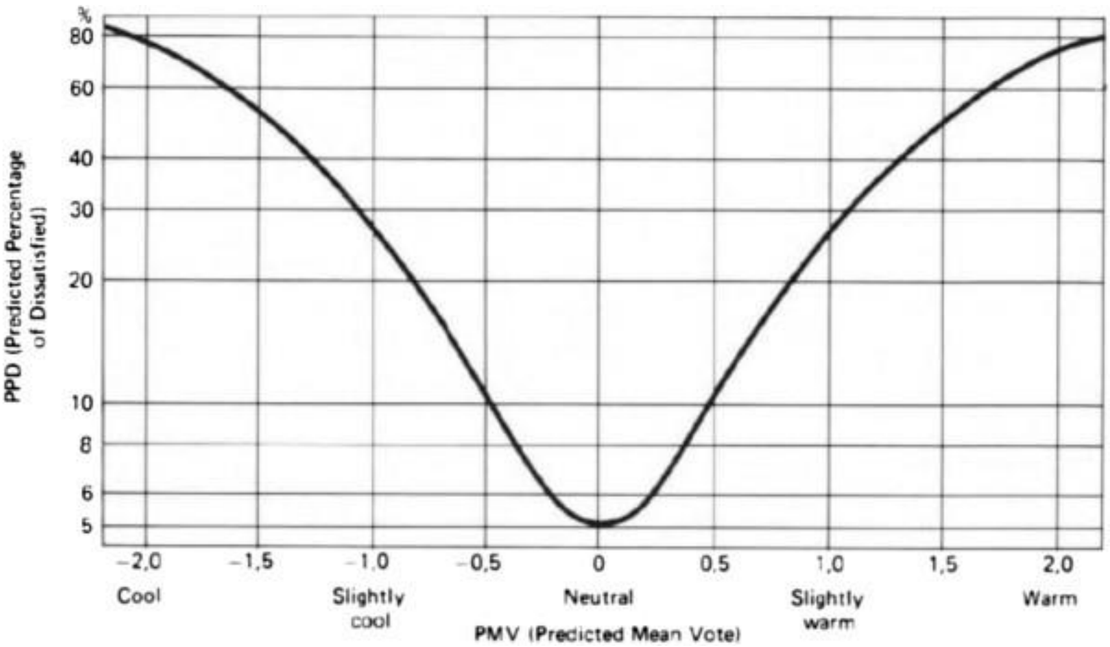
NOTE 1 metabolic unit = 1 met = 58.2 W/m²; 1 clothing unit = 1 clo = 0.155 m² · °C/W.

Figure 5-20 ISO 7730’s provided calculation for PMV for a specific environment

In order to compute PMV, the simulated temperature and airspeed velocity (i.e., the ASHRAE/ISO standards recommend to make an adaption for speeds above 0.2m/s) of a given environment are used as inputs. These variables, along with given inputs for clothing insulation, relative humidity, and mean radiative temperature provide the basis to calculate PMV.

Through PMV, we can predict the thermal sensation of a population, but this doesn't paint the whole picture. We also need to consider the level of satisfaction of the occupants in a space, to get a more holistic idea of if and how thermal comfort can be achieved. For this, Fanger developed another equation to relate the PMV to the predicted percentage of dissatisfied (PPD).

Once the PMV is calculated, the PPD, or index that establishes a quantitative prediction of the percentage of thermally dissatisfied occupants (i.e., too warm or too cold), can be determined. PPD essentially gives the percentage of people predicted to experience local discomfort. The main factors causing local discomfort are unwanted cooling or heating of an occupant's body. Common contributing factors are drafts, abnormally high vertical temperature differences between the ankles and head, and/or floor temperature.




December 2013Journal of Environmental Health Science and Engineering 11(1):32DOI:10.1186/2052-336X-11-32

Figure 5-21 The relationship between PMV and PPD

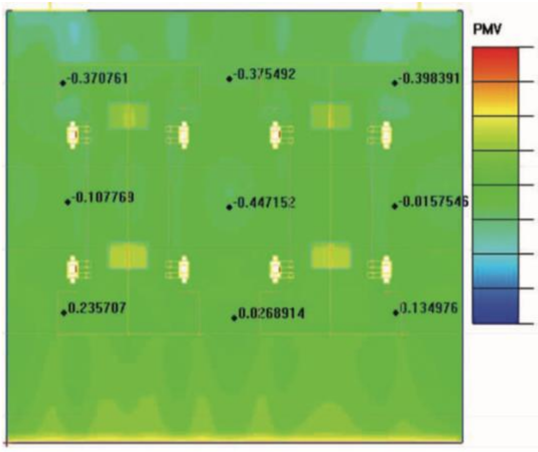


Figure 5-22 PMV at Y=1.1m plane of case A

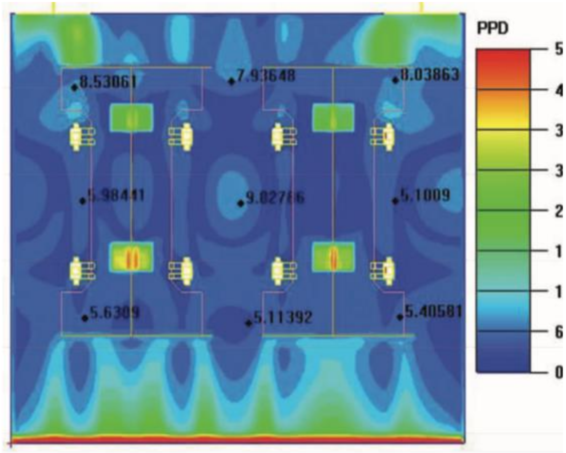


Figure 5-23 PPD at Y=1.1m plane of case A

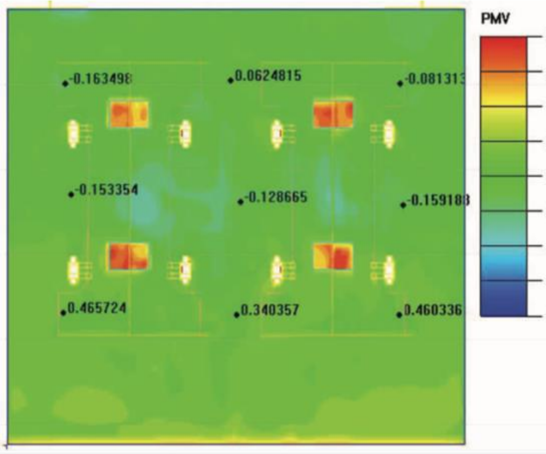


Figure 5-24 PMV at Y=1.1m plane of case B

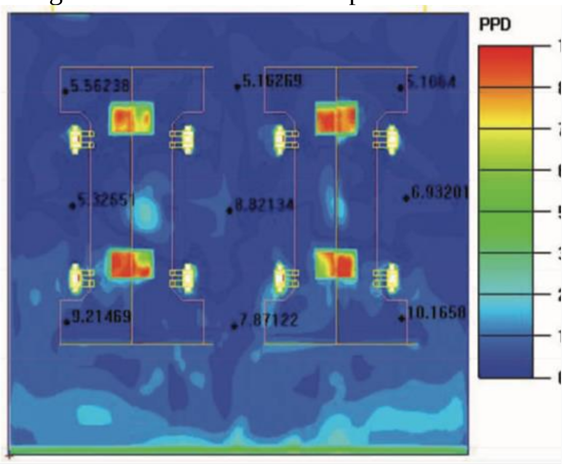


Figure 5-25 PPD at Y=1.1m plane of case A

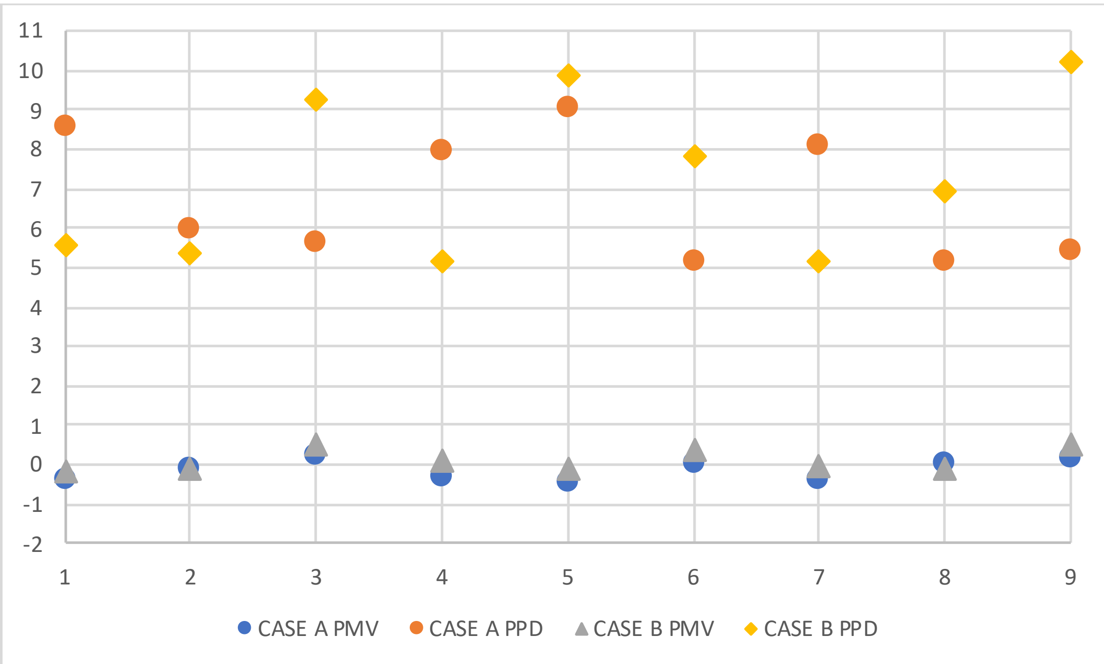



Figure 5-26 PMV,PPD at y=1.1m plane of Case A and Case B

The PMV and PPD of the 9 points uniformly distributed in the working area (from top to bottom and from left to right) in Figs.5-20，5-21 and 5-22，5-23 are summarized in Fig. 5-24, and the analysis shows that, under the action of scheme A, due to the gentle and uniform air supply, the cold air in the upper part of the room quickly exchanges heat with the indoor air, and when the airflow drops to about Y=1.1m (human breathing area) the PMV is -0.45- 0.24, human thermal sensation is moderate, PPD is between 5.1% andThe PMV was -0.04-0.61 with a moderate thermal sensation and the PPD was between 5% and 12.50%, with the PPD exceeding the IS0 standard of no more than 10%.

Numerical simulations and analyses of the velocity field, temperature field, air age, PMV and PPD were carried out for the office air conditioning system under both scenarios. The results show that: the air velocity in the working area of both scenarios can reach the requirements, the personnel do not feel the blowing air significantly, the airflow disturbance is smaller in scenario A compared to scenario B, the temperature fluctuation in the breathing area of the personnel is smaller, and the vertical temperature difference in other areas of the room is smaller. From the air age simulation data, it can be seen that the fresh air in Scenario A is delivered in a timely manner and the indoor pollutants are discharged in a timely manner, resulting in better air quality in the indoor work area. From the PMV values it can be seen that the thermal sensation of the personnel in both solutions is comfortable, while in solution B the dissatisfaction rate of the personnel in the local area has exceeded 10% and long stays should be avoided. It can be seen that the air outlet arrangement is one of the main factors affecting the distribution of airflow organisation, and various situations should be taken into account when selecting the air supply method.

## 5.5. Summary

From the analysis, it is clear that the temperature difference between the two scenarios is not significant at y=0.1m, and at y=1.1m the temperature of scenario b is significantly higher, although the room temperature shows a low temperature near the ground from time to time, but the comparison of the two scenarios is, we guess, due to the larger area of the room and the more violent disturbance of the indoor air, this point is also reflected in the velocity distribution diagram. In option b the airflow organisation is more fluid, with more vortices, and there are individual locations with higher air velocities and uneven airflow. The airflow is not as efficient as it could be in the room, and the airflow does not take away the heat from the equipment and the human body in time, making the temperature at the height of the human body in a sitting position slightly higher. In this regard, the temperature difference in option Bis larger.

During the flow of air from the air supply into the room, indoor pollutants are continuously mixed and the cleanliness of the air decreases, but due to the smooth airflow organisation in scheme a, the air exchange efficiency is better. In scheme b, however, the airflow disturbance area is relatively large, the replacement effect is slightly worse and the air quality is slightly poorer.

The results show that: the air velocity in the working area of both scenarios can reach the requirements, the personnel do not feel the blowing air significantly, the airflow disturbance is smaller in scenario A compared to scenario B, the temperature fluctuation in the breathing area of the personnel is smaller, and the vertical temperature difference in other areas of the room is smaller. From the air age simulation data, it can be seen that the fresh air in Scenario A is delivered in a timely manner and the indoor pollutants are discharged in a timely manner, resulting in better air quality in the indoor work area. From the PMV values it can be seen that the thermal sensation of the personnel in both solutions is comfortable, while in solution B the dissatisfaction rate of the personnel in the local area has exceeded 10% and long stays should be avoided. It can be seen that the air outlet arrangement is one of the main factors affecting the distribution of airflow organisation, and various situations should be taken into account when selecting the air supply method.

# CHAPTER 6
# CONCLUSION

## 6. Conclusion

Nowadays, people spend most of their time (70% -90%) indoors, and indoor air quality (IAQ) directly affects people's production, life and health, so IAQ is attracting more and more attention. With the development of air conditioning technology, research into the indoor air environment is already shifting from comfort to healthcare air conditioning. The requirements for human comfort are met by air-conditioning systems that regulate indoor environmental parameters or exploit the potential of natural energy sources. In particular, telecommuting has increased due to the COVID-19 pandemic of late, and this trend is expected to continue for a while. The ability of the virus to survive in the air depends on environmental conditions, especially ambient temperature and relative humidity are the most important parameters. A series of studies have shown that the dry environment seems to create an excellent environment for the growth and spread of influenza viruses.

In recent years, with the increasing capacity and speed of computers and the development of Computational Fluid Dynamics (CFD), CFD technology has been widely used in the field of HVAC. It becomes possible to simulate and predict the airflow organization, thermal comfort and optimal design of indoor air-conditioning systems based on CFD technology.
However, using CFD simulation method to model and simulate the humidity distribution field of indoor air environment is not comprehensively over world.

For this purpose, with the professional software Airpak 2.1 launched by Fluent, the temperature and humidity distribution are simulated. In order to test the feasible of CFD method, the simulation result was analyzed and checked by experiment data, and the results are validated by actual experiment. It is hoped that the research results can control of the decision-making of indoor temperature and humidity distribution.

This paper, firstly, expands the CFD calculation principle and method from the perspective of mathematical theory, and then determines the feasibility of this technology for indoor air temperature and humidity simulation based on this theory. Based on this method, in the thereinafter chapter, this paper carried out CFD simulation of laboratory temperature and humidity in combination with actual laboratory conditions. The model can realistically simulate the effect of ventilation on the condensation distribution. The ventilation method affects the condensation distribution and condensation sequence of the wall surface, so the wall condensation can be effectively controlled by properly arranging the position of the outlets.

For buildings affected by wet sources, ventilation with lower supply and upper back mode is better than other methods. Case 3 ventilation provides better air flow in the room and better air replacement in terms of air age, which prevents condensation from occurring inside the room. It can be inferred that when the wall of the ventilated room is added with insulation, replacing the material with good heat insulation and moisture resistance can also control the occurrence of condensation. However, there are still some important issues that need to be discussed in the future, including the following: The conclusions of this paper are based on small rooms and it remains to be seen whether similar conclusions can be drawn for other buildings, such as the problem of split-level air conditioning in tall spaces.

For split type air conditioner room, airflow organization distribution with the installation position of the air conditioner, indoor equipment placement and indoor personnel distribution, different airflow organization will form different velocity field and temperature field.

For office buildings, which are now mostly controlled by central air conditioning, the VRV plus fresh air air conditioning system and the primary return air conditioning system are also worth discussing with regard to the distribution of the indoor temperature and humidity fields and the indoor air quality.

In order to study the effect of different ventilation methods on the indoor humidity distribution and the formation of indoor condensation conditions, a simple physical model was established under the existing mathematical model, and CFD simulation was performed on the humidity distribution in each model under different ventilation methods using CFD technology. The results show that the air supply and return air outlets are on both sides of the room and the bottom-in and back-up method is the best way to control the indoor air humidity and greatly reduce the occurrence of dew condensation.

According to the simulation results, in the case of the room ventilation method, a small single apartment is taken as the research object, and the effect of the humidifier placement and height on the indoor humidity distribution in winter was studied. And get the result that at a height of 0.5 meters, condensation on the ground rarely occurs. 2) Within the scope of the nearby, the air just can hold limited capacity water vapor. The humidifier working efficiency was infected by the different position of humidifiers.

The humidity distribution in the room is related to the ventilation mode, and the flow pattern of the indoor airflow affects the distribution of indoor humidity. The room with the air supply and the return air outlets in the same direction, there is a difference in humidity on both sides of the room, and when the humidifier is used, the humidity on the other side of the room changes little. 4) The placement height of the humidifier also affects the indoor humidity distribution. When the humidifier is placed in a slightly higher position, the average humidity in the room will increase, and it will be easier to reach a comfortable humidity environment at a sleep height of 0.6 meters. But at the same time, there will be condensation of water near the humidifier.

The placement of the humidifier also has a great influence on the indoor humidity and humidity distribution. when the humidifier is placed beside the door, the humidifier does not improve the indoor humidity. When the humidifier is placed in the centre of the room, the humidity in the room is evenly distributed. When the humidifier is placed close to the wall, condensation occurs near the ground due to the low wall temperature.

From this simulation, it can be concluded that in the dry winter season, by arranging the position of the indoor vents and placing the humidifier reasonably, the humidity can reach a comfortable range while reducing the possibility of condensation.

At present in Japan, due to the internal activities of human and the heat dissipation of various equipment in office buildings, more and more office buildings use air-conditioning for cooling in winter, and due to the cooling system, the supply air cannot be humidified during cooling. Indoor humidity conditions are not comfortable. This article simulates the indoor temperature and humidity of the office building refrigeration system in winter. It is hoped that the research results can guide the control decision of actual indoor temperature and humidity.

From the analysis, it is clear that the temperature difference between the two scenarios is not significant at y=0.1m, and at y=1.1m the temperature of scenario b is significantly higher, although the room temperature shows a low temperature near the ground from time to time, but the comparison of the two scenarios is, we guess, due to the larger area of the room and the more violent disturbance of the indoor air, this point is also reflected in the velocity distribution diagram. In option b the airflow organisation is more fluid, with more vortices, and there are individual locations with higher air velocities and uneven airflow. The airflow is not as efficient as it could be in the room, and the airflow does not take away the heat from the equipment and the human body in time, making the temperature at the height of the human body in a sitting position slightly higher. In this regard, the temperature difference VRV multi-connector plus fresh air air-conditioning system is larger.

During the flow of air from the air supply into the room, indoor pollutants are continuously mixed and the cleanliness of the air decreases, but due to the smooth airflow organisation in scheme a, the air exchange efficiency is better. In VRV multi-connector plus fresh air air-conditioning system, however, the airflow disturbance area is relatively large, the replacement effect is slightly worse and the air quality is slightly poorer.

The results show that: the air velocity in the working area of both scenarios can reach the requirements, the personnel do not feel the blowing air significantly, the airflow disturbance is smaller in all-air primary return air system compared to VRV multi-connector plus fresh air air-conditioning system, the temperature fluctuation in the breathing area of the personnel is smaller, and the vertical temperature difference in other areas of the room is smaller. From the air age simulation data, it can be seen that the fresh air in all-air primary return air system s delivered in a timely manner and the indoor pollutants are discharged in a timely manner, resulting in better air quality in the indoor work area. From the PMV values it can be seen that the thermal sensation of the personnel in both solutions is comfortable, while in VRV multi-connector plus fresh air air-conditioning system the dissatisfaction rate of the personnel in the local area has exceeded 10% and long stays should be avoided. It can be seen that the air outlet arrangement is one of the main factors affecting the distribution of airflow organisation, and various situations should be taken into account when selecting the air supply method.

## References


[1] Joyce，W. 2003. Solutions for Energy Security & Facility Management Challenges，1st Ed. p460.Fairmont Press.

[2] Yueling,M., Yadong,Z., Jiangtao,L., Xiaotao,H., Bo,W., Shihua,F., Jun,Y., Jingping,N. ,Ji,Z., Bin,L., Effects of temperature variation and humidity on the death of COVID-19 in Wuhan, China. *Sci Total Environ.* Volume 724.

[3] G.L. Tuve.1954. *Air velocities in ventilating jets*. 1ST Ed. pp.261～ 282. Transactions Amer Soc Heating & Air-Conditioning Engr.

[4] Howarth, A. 1983. Temperature distributions and air movements in rooms with a convective heat source Ph.D.Thesis, England, University of Manchester Institute of Science & Technology,UK.

[5] Guoqiang,Z., Lin,Y., Haghighat,F. 2000. Indoor airflow simulation methods comparison and a new Zonal model approach study. *Applied Journal of Basic and Engineering Sciences,2000, 8(3): 291-300.*

[6] Nielsen, P. V. 1974. Flow in air conditioned rooms. Ph.D.Thesis, Denmark, Technical University of Denmark, Copenhagen.

[7] Markatos,N.C.，Kirkcaldy,D. 1983. Analysis and computation of three-dimensional，transient flow and combustion through granulated propellants. *International Journal of Heat & Mass Transfer*. 26(7)：1037-1053.

[8] King,A.R.，Kronfalt,M.，Clements,R.F.1993. Stratified air conditioning of large spaces with high heat loads. *Australian Refrigeration, Air conditioning & Heating* 47n2, 24-29..

[9] Lee K,Jiang Z,Chen. 2009. Air distribution effectiveness with stratified air distribution systems. *ASHRAE Transactions*，115(2)：322-333

[10] Zítek.P., Vyhlídal.T., Simeunović.G., Nováková. L., Čížek. J. 2010.Novel personalized and humidified air supply for airliner passengers. *Build and Environment* 45: 2345–2353.

[11] Yawei,H.2016.Airpak-based airflow organization of large stadiums for energy-saving design and Design and energy-saving operation of large stadiums based on Airpak. Ph.D.Thesis,Jilin University of Architecture, Changchun.

[12] Wu.D, Chen.W, Liang.J.C. 2006.Experimental study of indoor temperature field and temperature drop rate during cooling operation of air conditioners. *Building Science*, 2006(5).53-58.

[13] Li ,Y., Yu ,W., H, Wang. F,L. 2017. Numerical simulation of the thermal environment in the office of a cabinet air conditioner in summer based on Airpak Numerical simulation analysis of thermal environment in the office of a cabinet air conditioner in summer[J]. *Journal of Northern University of Technology Journal*, 29(2): 122-130.

[14] Shouyuan,H., Fujun, Z., Gang,L. 2011. Airpak-based summer air conditioning indoor thermal environment based on Airpak . *Journal of Hunan University of Science and Technology (Natural ScienceEdition),* 26(2): 11-17.

[15] Qiang,T., Xiande,H. 2017.Numerical simulation of indoor air environment based on Airpak software. *Journal of Jingdezhen College*, 32(3): 30-33.

[16] Caixia,L., Shenghua,Z., Ruhui,Y. 2012.Airpak-based indoor air quality analysis. *Refrigeration and Air Conditioning (Sichuan)*, (4): 381-384.

[17] Launder,B,E., Rodi,W. 1983. The turbulent wall jet-measurements and modeling. *Annual Review of Fluid Mechanics*.15.429-459.

[18] Rahmood,A., Yaghoubi. 1995. Three-dimensional numerical simulation of air contamination dispersal in a room. *ASHRAE Technical Data Bulletin,* 59-68.

[19] Zhixin.M. 2017. Investigation and measurement of residential winter wet environment in Shanghai area. *Cooling and Air Conditioning (Sichuan),* 2017 (1): 32-37.

[20] Lin,L. 2012.Analysis of condensation on the roof of large space buildings caused by the migration of humid air in hot and humid areas. *HVAC,* 42 (12): 109-13.

[21] Xingguo,G.,Youming.C. 2015. Hot and humid climate Condensation analysis of internal walls. *Civil Engineering and Environmental Engineering,* 3 (5): 121-124.

[22] Yiding,W. 2017. Numerical model of air distribution in air-conditioned rooms under different air supply speeds Proposed. *Building Energy Conservation,* 2017 (5): 3-36.

[23] Shui,Y. 2016.Analysis of the heat and moisture characteristics of the inner wall of the underground outdoor wall in severe cold areas.*Building Energy Conservation* 2016(1): 48-52.

[24] Fang,T.,Feng,H.,Wu,B.2011.Indoor ventilation simulation in green building based on Airpak. International conference on computer distributed control and intelligent environmental monitoring. IEEE computer society,2011,848-851

[25] Shui,Y. 2010.Numerical Simulation Analysis of Heat and Moisture Coupling in Envelope. *Building Energy Conservation* (7): 68-71.

[26] Jiefeng,Y. 2017.Research on the influence of different turbulence models and difference schemes on numerical simulation and prediction of typical flow field at low Reynolds number. *Refrigeration and Air Conditioning (Sichuan)*,(5): 1-17.

[27] Daisei Yamano, Kosaku Ueda, Mitsunori Sato, 2019.1st Ed.Grade Architect Department Exam Essentials Check 2019 Edition. P71.

[28] Xiaojun,M. 2012.Distribution and application of temperature, humidity and pollutants in ventilated and air-conditioned rooms. Ph.D.Thesis ,Tsinghua University, Beijing.

[29] Chao,L., Hong,L., Baizhan,L.2018. Seasonal effect of humidity on human comfort in a hot summer/cold winter zone in China. *Indoor and Built Environment,* vol. 28(2): 264-277

[30] Yokasta,G.F., Letzai,R.V., Jesús,C.R. 2018.Indoor temperature and relative humidity assessment of three construction systems for Dominican social housing in different micro-climates: A modelling study. *Indoor and Built Environment,* vol. 28(5): 693-710.

[31] H. Li, G. Gong, C. Xu, H. Su, Z. Cao, M. Zhou, C. W. F. Yu. 2013.Thermal and humid environment and moisture condensation characteristics of cold surfaces. *Indoor and Built Environment,* vol. 23(3): 474-484.

[32] Z.Q. Liu，G. Li，C.X. Zheng．The simulation and calculation of indoor thermal environment and the PPD evaluation index affected by three typical air-conditioning supply modes[J]．*HVAC&R Research*，2014，20：343-350

[33] P. Taffé． A qualitative response model of thermal comfort[J]． *Building and Environment*，1997，32(2)：115-121

[34] P.O. Fanger,Thermal comfort-analysis and application in environment engineering [M]． Copenhagen：Danish *Technology Press*，1970．

[35] H. Zhang，H.C. Huizenga，E. Arens．Considering individual physiological differences in a human thermal model[J]．*Journal of Thermal Biology*，2001，26：401-408．

[36] S. Heidari，S. Sharples．A comparative analysis of short-term and long-term thermal comfort surveys in Iran[J]．*Energy and Buildings*，2002，34：607-614．

[37] 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[S]．Geneva，International Organization for Standardization，2005．

[38] ASHRAE，ANSI/ASHRAE Standard 55-2010：Thermal environment conditions for human occupancy [S]．Atlanta，GA，American Society of Heating， Ventilating and Air-Conditioning Engineers，Inc.，2011．

[39] J. Toftum A.S. Jorgensen，P.O. Fanger．Upper limits for indoor air humidity to avoid uncomfortably humid skin [J] Energy and Buildings，1998，28：1–13．

[40] R.G. Nevins．Temperature-humidity chart for thermal comfort of seated persons [J]． ASHRAE Transactions，1966，72：283-291．

[41] Mc Nall，P.E. Jaax，F.H. Rohles，R.G. Nevins，W.E. Springer．Thermal comfort (thermally neutral) conditions for three levels of activity [J]．ASHRAE Transactions，1967，73 (1)：1-14．

[42] L. Berglund．Comfort criteria in a low humidity environment [R]．Final Report to Electric Power Research Institute, Palo Alto，Calif，1989．

[43] L. Berglund．Comfort criteria and humidity standards [C]．Proceedings of the Pan-Pacific Symposium on Building and Urban Environmental Conditioning in Asia，March 16-18． Nagoya，Japan，1995，2：369-382．

[44] D.A. Mc Intyre， I.D. Griffths． Subjective responses to atmospheric humidity [J]． Environmental Research，1975，9：66-75．

[45] T. Xu，E. Arens，F. Bauman．The effects of high- level air humidity on subjective perception of comfort [C]．Proceedings of the 2nd International Symposium on Heating， Ventilation and Air Conditioning, 25-27 September，Beijing,China，1995，1：81-91．

[46] Marc E. Fountain, Edward Arens, Tengfang Xu, Fred S. Bauman, Masayuki O guru. An investigation of thermal comfort at high humidities [J]. ASHRAE Transactions, 1999, 105 (2): 94-103.
[47] R. Nevins. Effeet of change in ambient temperature and level of humidity on comfort and thermal sensation[J]. ASHRAE Transactions, 1975, 81(l): 64-77.
[48] J. Toftum, A.S. Jorgensen, P.O. Fanger. Upper limits of air humidity for preventing warm respiratory diseomfort[J]. Energy and Buildings, 1998(28): 15-23.
[49] L.G. Berglund. Pereeived air quality and the thermal environment[J]. IAQ, 1989, 89: 93-99.
[50] Zhang Quan, Yuan Hao. Study on the influence of indoor human movement processes on thermal comfort[J].The Journal of Hunan University (Natural Sciences)2007(8).1- 5
[51] Sun Yong, Jin Xinqiao, Du Zhimin, Sun Jinlong. Analysis of indoor humidity control in variable air volume air conditioning system[J].Journal of Shanghai Jiaotong University.2005(2): 216-219
[52] Pu L, Xu J, Yu F, Gao C Y, Li Y Z. Numerical simulation of humidity fields in electrical equipment environmental laboratories andoptimization[J].The Journal of Xi'an University of Architecture and Technolog (Natural Science Edition) , 2007(5): 716-720.
[53] Wang Yizhong, Hu Chunyuan, Chen Shaohui, Zhang Ping, Huang Huafang, Huang Guoquan, Liu Xu. Numerical simulation of the temperature and humidity fields in a refrigerated trucktemperature and humidity fields in a refrigerated truck[J].The Preservation and Processing.2010(3): 26-29.
[54] Liu G, Wang D-P. Influence of air moisture content on temperature distribution in train cars [J].The Science and Technology Information Development and Economy.2007(31): 138-140.
[55] Zhai Yongchao. Experimental study on the effect of air flow on human thermal comfort under humid heat environment [D] . Guangzhou: South China University of Science and Technology.2013.
[56] Ma Xiaojun. Research on temperature, humidity and pollutant distribution patterns in ventilated and air-conditioned rooms and their applications[D].The Beijing: Tsinghua University, Beijing.2012.
[57] Chen YJ. Study on the control of relative humidity in air conditioning systems under different conditions[D].The Xi'an: Xi'an University of Architecture and Technology2008.
[58] Mao Yan. Study on climate adaptation for human thermal comfort [D] , Xi'an: Xi'an University of Architecture and Technology.2006
[59] ASHRAE. Thermal environment conditions for human occupancy (ASHRAE Standard 55-1992) [S]. Atlanta: ASHRAE, 1992.
[60] Luo Yuan, Xu Culture. Influence of average radiation temperature on human thermal comfort in sleeping environments[J]. The Refrigeration Technology.2010(4): 52-56.
[61] Huang Chen. Built Environment[M].The Beijing: Machinery Industry Press.2005
[62] Zheng,TQ. The effect of activity level on human thermal sensation[D],Chongqing: Chongqing University, China.2014.
[63] Han Yi. Analysis of indoor air quality in buildings[J].The Building Science.2014(7): A202.
[64] Cui T, Chen SQ. Research progress of indoor air quality evaluation methods and standards[J].The Refrigeration and Air Conditioning.2004(2) :63-67
[65] M. Sandberg, M. Sjoberg. The use of moments for assessing air quality in ventilated rooms[J]. Building and Environment, 1983, 18(4): 187-197.
[66] A.K. Melikov, R. Cermak, M. Milan. Personalized ventilation: evaluation of different air terminal devices[J]. Energy and Buildings, 2002, 34: 826-836.
[67] Ma Renmin. Ventilation effectiveness and indoor air quality[J]. HVAC.2000.30(5).20-23.
[68] Tian Huiling. Numerical study of the effect of airflow organization on indoor air quality[D]. Henan: Henan University of Technology.2009.
[69] Sun Jining. Numerical simulation of convective heat transfer: a tutorial for basic applications[M]. Beijing: National Defense Industry Press, 2010.
[70] Xie Longhan, Zhao Xinyu.ANSYS CFX Fluid Analysis and Simulation (2nd ed.). Version 2)[M]. Beijing, 2013.

[71] Wu Zhenjing. Study on the characteristics of indoor fine particle flow influenced by air supply method[D]. Tianjin: Tianjin University of Commerce.2015.
[72] Yu Y. FLUENT Introduction and Advanced Tutorial[M]. Beijing: Beijing University of Technology Press, 2008.
[73] Li Ge. Study on the influence of individualized air supply end methods on the thermal environment of workstation area at desktop[D]. Tianjin: Tianjin University of CommerceTianjin: Tianjin University of Commerce, 2015.
[74] Xie LH, Zhao XY, Zhang JM.ANSYS CFX Fluid Analysis and Simulation (Version 1) Version 1)[M]. Beijing.Electronic Industry Press, 2012.
[75] Song Danping. Experimental study and numerical simulation of thermal stratification height in natural displacement ventilation system[D]. Tianjin.Tianjin University of Commerce, 2008.
[76] Wang Junshun. Analysis and research on the effect of desktop air supply on human thermal comfort [D]. Hunan: Hunan University, 2008.
[77] Tao Wenquan. Numerical Heat Transfer(2nd ed.) [M]. Xi'an: Xi'an Jiaotong University Press.2001.
[78] Zhao Yuxin.FLUENT Tutorial, Institute of Aerospace, National University of Defense Technology.
[79] CFX Ltd．CFX handbook [M]．United Kingdom：Didcot Oxfordshire，2009．
[80] S.A. Mumma．Chilled ceilings in parallel with dedicated outdoor air systems：Addressing the concerns of condensation，capacity，and cost．ASHRAE Transactions，2002，108(2)：220-231．
[81] L. Bellia，F. Minichiello．A simple evaluator of building envelope moisture condensa- tion according to an European standard. Building and Environment，2003，38：457-468．
[82] J. Liu，H. Aizawa，H. Yoshino．CFD prediction of surface condensation on walls and its experimental validation．Building and Environment，2004，39：905-911．
[83] ASHRAE．ASHRAE handbook：heating，ventilating，and air-conditioning systems and equipment．Atlanta，ASHRAE Inc，2004．
[84] Bellia L., Minichiello.F., 2003. A simple evaluator of building envelope moisture condensation according to an European standard.Building and Environment 38: 457-468.
[85] Braconnier, R., Bonthoux, F. 2007. A numerical method of reconstructing the pollutant concentration
[86] field in a ventilated room, Annals of Occupational Hygiene, 51, p311-325
[87] Cai H., Li X. T., 2008. Balancing Indoor Environment Quality and Energy Use with a Human-oriented Evaluation Method of Indoor Air Distribution，Proceedings of 1st International Conference on Building Energy and Environment (COBEE 2008), pp. 2474-2481.
[88] Cermak R., M. Majar, A.K. Melikov. 2002. Improved quality of inhaled air with personalized ventilation, in: Proceedings of Indoor Air 2002, vol 3, Monterey, California: 1054-1059.
[89] Chad B, Dorgan P E, et al. 1998. Health and productivity benefits of improved indoor air quality. ASHRAE Transactions, 104(1): 658-665
[90] Chen Q. 1988. Indoor Airflow, Air Quality and Energy Consumption of Buildings. Netherlands: Krips Repro Meppel.
[91] Chen Q Y and Xu W. 1998. A zero-equation turbulence model for indoor air flow simulation. Energy and Buildings, 28: 137-144
[92] Chen, Q. Glicksman, L.R. 2003. System Performance Evaluation and Design Guidelines for Displacement Ventilation, 129 pages, ISBN 1-931862-42-7, ASHRAE, Atlanta, GA.
[93] Fang L., Clausen G., Fanger P.O. 1999. Impact of Temperature and Humidity on chemical and sensory emissions from building materials. Indoor Air 9 :193-201.
[94] Fanger P. O., 1970. Thermal comfort. Analysis and applications in environmental engineering.
[95] Copenhagen, Danish Technical Press.
[96] Fanger PO and Olesen B W. 2004. Indoor air more important for human health than outdoor air. Bridging from technology to society. Copenhagen: Technical University of Denmark, 66-73
[97] Fanger, P.O. 2001. Human requirements in future air-conditioned environments, International Journal of Refrigeration-Revue Internationale Du Froid, 24, 148-153.

[98] Franklin G.F., Powell J.D., Emami-Naeini A. 2002. Feedback control of dynamic systems (5thEd.). Pearson prentice hall. New Jersey. 928p.
[99] Green G.H. 1982., Positive and negative effects of building humidification, ASHRAE Transaction, 88(1):1049-1061
[100] Hand D.W. 1998. Effects of atmospheric humidity on greenhouse crops, Acta Horticulturae 229:143–158.
[101] Heiselberg P. 1996. Room air and contaminant distribution in mixing ventilation [J]. ASHRAE Transactions, 102 (2): 332-339.
[102] Hippert H.S., Pedreira C.E., Souza, R.C. 2001. Neural networks for short-term load forecasting: a review and evaluation, IEEE Transactions on Power Systems, Volume: 16 Issue:1, page(s): 44 – 55
[103] Hiyama K., Kato S., Ishida Y. 2010. Thermal simulation: Response factor analysis using three-dimensional CFD in the simulation of air conditioning control. Building Simulation. Volume 3, Number 3, 195-203
[104] Huang H., Kato S., Hu R., Ishida Y., 2011. Development of new indices to assess the contribution of moisture sources to indoor humidity and application to optimization design: Proposal of CRI(H) and a transient simulation for the prediction of indoor humidity, Building and Environment, 46: 1817-1826
[105] Huang H Y, Haghighat F. 2001. An integrated IAQ model for prediction of VOC emissions and sinks by building materials. Proceedings of the 4th IAQVEC,Changsha, pp367-374
[106] ISO 7730. 1994. International Standards Organization. Moderate Thermal Environments: Determination of the PMV and PPD Indices and Specification of the Conditions for Thermal Comfort
[107] Jaakkola J.J.K., Heinonen O.P., Seppanen O. 1989. Sick building syndrome, sensation of dryness and thermal comfort in relation to room temperature in an office building: Need for individual control of temperature. Environment International. Volume 15, Issues 1–6, Pages 163–168
[108] Jones P. J., Whittle G. E. 1992. Computational fluid dynamics for building air flow prediction-current status and capabilities. Building and Environment. Vol. 27(3): 321-328.
[109] Kalogirou S. A. 2000. Applications of artificial neural-networks for energy systems，Applied Energy，Volume 67, Issues 1-2, Pages 17-35
[110] Kato S., Murakami S., Kobayashi H. 1994a. New Scale For Evaluating Ventilation Efficiency as Affected by Supply and Exhaust Openings Based on Spatial Distribution of Contaminant，12th International Symposium on Contamination Control, The Japan Air Cleaning Association: 341-348
[111] Kato S., Murakami S., Kobayashi. H. 1994b. New scales for assessing contribution of heat sources and sinks to temperature distributions in room by means of numerical simulation, ROOMVENT'94, 539-557
[112] Kitagawa K., Komoda N., Hayano H., Tanabe S. 1999. Effect of humidity and small air movement on thermal comfort under a radiant cooling ceiling by subjective experiments. Energy and Buildings 30:185–193.

## Acknowledgement

My study at the University of Kitakyushu will soon come to an end and, at the completion of my thesis; I wish to express my sincere appreciation to all those who have offered me invaluable help during the four years study. The successful completion of this thesis is inseparable from the support and help of many people. I would like to take this precious opportunity to express my sincerest gratitude.

First of all, My deepest gratitude goes first and foremost to Professor Ryu Yuji, my supervisor,. From the initial conception of the report, the rationality of the outline design and the later revisions, he helped me a lot. At the same time, he has also given me a lot of inspiration in my usual academic and daily life, which has benefited me a lot.

Secondly, I am also greatly indebted to the professors and teachers at the Faculty of International Environmental Engineering of the University of Kitakyushu: Professor Gao Weijun, Professor Bart, Professor Fukuda, who have instructed and helped me a lot in the past four years, their professional knowledge and patient teaching have not only given me academic guidance, but also guided my project progress and direction also taught me how to be a responsible researcher and enriched my life.

I also want to deeply thank my lovely classmates, thank you for your active cooperation and outstanding performance in my research. It is your strong support that enabled my research to be implemented and successfully completed. Of course, I want to thank my friends. Their enthusiasm and understanding, support and help are the precious wealth of my life.

Finally, I would also like to thank my family for their support and understanding during my Ph.D. years. They have always been my strongest backing. Only with their selfless support, concern and love, can I overcome those difficulties and pursue my study till now. Their loving considerations and helps are the source of my strength.