

Computer Simulated Person to Assess Micro-Environment Heterogeneity Indoors

朴, 炫奎

<https://hdl.handle.net/2324/7329514>

出版情報 : Kyushu University, 2024, 博士 (学術), 課程博士
バージョン :
権利関係 :



**Computer Simulated Person to Assess
Micro-Environment Heterogeneity Indoors**

PARK Hyun-gyu

August 2024

A dissertation for Doctor Degree

**Computer Simulated Person to Assess
Micro-Environment Heterogeneity Indoors**

Interdisciplinary Graduate School of Engineering Sciences

Kyushu University

PARK Hyun-gyu

Supervisor

Prof. Kazuhide ITO

August 2024

This page intentionally left blank

ACKNOWLEDGEMENTS

First and foremost, I extend my deepest gratitude to my supervisor, Prof. Kazuhide Ito, who has been an exceptional guide and mentor throughout my doctoral journey. His insights and feedback were invaluable throughout this process. His profound expertise and insightful guidance have been pivotal in my academic and personal growth. His commitment to excellence and his patient, detailed feedback on my work have not only shaped this dissertation but have also prepared me for a future in scientific research. I am especially thankful for the significant time he dedicated to mentoring me, providing numerous valuable suggestions and insightful feedback that were crucial in advancing my research. Additionally, I am truly grateful for the generous opportunities he provided that allowed me to gain diverse experiences and form broader community relationships at international conferences.

I am profoundly grateful to Assoc. Prof. Sung-Jun Yoo, who has provided endless help, reviews, advice, and discussions from the beginning to the end of my doctoral journey. Words cannot adequately express my thanks. The knowledge and experience he shared, as well as his kind and passionate personality, have deeply impressed me throughout this research journey. I am grateful to Assist. Prof. Kazuki Kuga for providing invaluable resources that is essential for my research, as well as for his insightful comments that significantly enhanced my work.

I am also deeply thankful to members of my doctoral cohort, Dr. Khoa and Dr. Ken, for their constant help and advice throughout my studies. The three years I spent with you have been enjoyable and a great fortune for me.

My doctoral program would not have been possible without the financial supports providing tuition and stipend by Q-Energy Fellowship's scholarship. I am very grateful for the valuable opportunity offered by the Japanese Government.

I would also like to express my deepest gratitude to Prof. Janghoo Seo of Kookmin University in Korea. He provided kind guidance from my master's program and helped me navigate my way to studying in Japan, offering various advice during my time abroad.

I would also like to express my gratitude to Dr. Seong-Hyun Park for always being generous with his advice.

I extend my special thanks to my family for their support and encouragement and love throughout my whole life. Thanks to them, I am who I am today, and I am always grateful and filled with love.

ABSTRACT

There is a growing demand for more advanced and diverse computer-simulated persons (CSPs) to accurately evaluate indoor environmental quality and analyze the interactions between environmental factors and the human body. For this purpose, it is essential not only to replicate the human shape but also to accurately reproduce human physiological functions which can affect the indoor environment. These physiological functions and their interactions with the indoor environment are critical for human risk assessment which serve as the basis for the design of ventilation systems. Therefore, this dissertation aims to develop a CSP with precise human geometric shapes and integrate three key physiological functions: thermoregulation, breathing, and micro-movement (movement to maintain human posture).

First, based on the ANDI thermal manikin, which has been widely used in experimental studies targeting indoor environments and the surrounding human body, a CSP and a heterogeneous clothing model were established through precise 3D scanning. To replicate the skin temperature and thermal functions of the ANDI thermal manikin in numerical analysis, the Fiala multi-node thermoregulation model was integrated into the CSP using a sub-routine program. Subsequently, the thermal prediction accuracy of the CSP was validated by comparing the results of chamber experiments using the ANDI manikin with the CSP-CFD coupled analysis results. Second, to replicate human breathing function of the developed CSP, a nasal cavity model, derived from CT data of the human respiratory system, was integrated and a transient breathing profile was utilized. Third, a single-mass Inverted Pendulum Model (IPM) was applied to the CSP to replicate the micro-movements that continuously occur in a stationary standing human to

maintain posture. Additionally, the impact of these movements on the formation of the Breathing Zone (BZ) and on the immediate surrounding of the human body was evaluated through a CSP-CFD coupled analysis. The BZs were evaluated using the Scale for Ventilation Efficiency (SVE) 4 and 5, corresponding to exhalation and inhalation modes, respectively. Through this analysis, it was found that the micro-movements of the CSP had a minimal impact on the area of the thermal plume formed around the body and also on the convective heat transfer coefficient (CHTC) of the CSP's skin, but they significantly altered the BZ. These findings emphasize the importance of incorporating micro-movements in CSP simulations for accurately assessing breathing air quality. However, the increased computational demands of replicating these detailed human interactions suggest the need for a balanced approach between accuracy and computational efficiency in studies where the detailed impact of human micro-movement on the indoor climate is not the primary focus. The summaries of the chapter contents of this dissertation are as follows:

Chapter 1 introduces the background and purpose of the dissertation, setting the stage for the detailed analyses that follow.

Chapter 2 outlines the methodology employed in the dissertation, focusing on the computational fluid dynamics (CFD) technique used, including turbulence, radiation, and passive scalar transport modeling.

Chapter 3 discusses the process of establishing the geometry of the CSP based on 3D scanning data of the target model which is ANDI thermal manikin. This section also illustrates the process of replicating the actual shape of clothing and how to integrate with the CSP. Additionally, this

chapter introduces the Fiala multi-node thermoregulation model that controls the thermal functions and skin temperature of the ANDI thermal manikin in response to the surrounding environment and discusses its integration with the CSP-CFD coupled analysis.

Chapter 4 focuses on validating the thermal prediction accuracy of the developed CSP when coupled with CFD simulation. To achieve this, the chamber experimental environment using the actual ANDI thermal manikin was replicated through CFD simulation, and a comparison of skin temperatures between the thermal manikin and CSP was conducted. The CSP analysis results reproduced the experimental results obtained using the thermal manikin with acceptable accuracy, and the root mean square deviation (RMSD) of the predicted average skin temperature was less than 0.3 in all cases. Furthermore, the importance of detailed clothing geometry was highlighted by confirming that the thickness of the ventilation layer between the skin and clothing has a significant effect on the thermal performance. However, in the cases with the seated CSP, the Pearson correlation coefficient did not show sufficient consistency, which necessitates further analysis to enhance reliability.

Chapter 5 explores the integration of a nasal cavity model into the developed CSP, replicating the vital human function of breathing. This chapter introduces a transient breathing cycle model, simulating variable breathing rates over time to facilitate dynamic numerical analysis. Additionally, this chapter analyzes the BZ distribution using Scale for Ventilation Efficiency (SVE) 4 and 5, which are crucial for evaluating contaminant transfer and assessing human exposure.

Chapter 6 discusses micro-movements resulting from postural sway and introduces the single-mass Inverted Pendulum Model (IPM) to replicate these movements. The chapter presents the results of CFD analysis that explores how these micro-movements influence the formation of the CSP's BZ and affect the general indoor climate. In the case where angular movement (single-mass IPM) was applied, while the distribution of exhaled air remained generally consistent, distinct differences were observed in the distribution of inhaled air compared to the case with no movement. In a representative case with a maximum angular velocity of $4^\circ/\text{s}$, the BZ's horizontal length expanded by 12 cm, and the vertical extent of the SVE5>10% distribution contracted by 25 cm compared to the stationary case. Notably, linear movement in the mediolateral (ML) direction resulted in a horizontal expansion of the inhaled air distribution. Additionally, it was confirmed that the micro-movements of the CSP have minimal impact on the distribution of the thermal plume surrounding CSP and on the convective heat transfer coefficient (CHTC) of the CSP skin surface. The findings of this study suggest that replicating human micro-movements has minimal impact on general indoor climate analysis but significantly improves the accuracy of predicting air quality in breathing zones or assessing inhalation exposure to contaminants.

In Chapter 7, the outcomes and achievements obtained from each chapter are summarized and described. Additionally, this chapter outlines the future tasks planned for this research.

KEYWORDS: Computational fluid dynamics (CFD), Computer simulated person (CSP), Clothing model, Human physiological function, Thermoregulation model, Breathing, Postural sway, Micro-movement, Inverted pendulum model (IPM), Transient breathing cycle, Breathing zone (BZ), Scale for ventilation efficiency (SVE) 4 and 5, Reynold-averaged navier stokes equation (RANS)

TABLE OF CONTENTS

CHAPTER 1: Introduction

1.1. Research bckground and prpose	1
1.2. Composition of dssertation	5
REFERENCES.....	8

CHAPTER 2: Outline of Numerical Analysis

2.1. Introduction.....	13
2.2. Numerical analysis of flow field, temperature field, and humidity field.....	14
2.2.1. Fundamental equations for fluid simulation	14
2.2.2. Nondimensionalization of the equation system	15
2.2.3. Analysis of equation systems through averaging and ensemble averaging	17
2.3. Turbulence model.....	19
2.3.1. Standard k- ϵ model	19
2.3.2. SST k- ω model.....	24
2.4. Surface to Surface (S2S) model	27
2.5. Passive scalar transport equation	32
REFERENCES.....	33

CHAPTER 3: CSP Integrated with a Multi-node Thermoregulation Model

3.1. Introduction.....	34
3.2. Overview of the numerical human model.....	34
3.3. Establishment process of CSP's geometry and clothing model.....	35
3.4. Fiala thermoregulation model	37
3.4.1. Body construction	37
3.4.2. Heat conduction	38

3.4.3. Metabolism.....	39
3.4.4. Blood circulation.....	40
3.4.5. Numerical methods	43
3.4.6. Core and surface temperature	45
3.5. Implementation of thermoregulation model through CSP-CFD coupled analysis	46
REFERENCES.....	49

CHAPTER 4: Validation of CSP’s Thermal Prediction Accuracy

4.1. Introduction.....	50
4.2. Chamber experiment using ANDI thermal manikin	50
4.3. CSP-CFD coupled analysis corresponding to chamber experiment	51
4.4. CFD analysis results and discussion	54
4.4.1. Airflow and temperature distribution.....	54
4.4.2. Clothing impact.....	57
4.4.3. Comparison of skin temperature between CSP and thermal manikin.....	62
4.5. Limitation.....	68
4.6. Conclusion	68
REFERENCES.....	69

CHAPTER 5: Human Breathing

5.1. Introduction.....	70
5.2. Integration of CSP with airway model.....	70
5.3. Transient breathing model.....	71
5.4. CFD analysis and validations.....	73
5.5. Scale for Ventilation Efficiency (SVE) 4 and 5 for analyzing the BZ	76
REFERENCES.....	79

CHAPTER 6: Micro-movement

6.1. Introduction.....	80
6.2. Review of previous research on human movement	80
6.3. CFD analysis for the effect of human micro-movement on BZ formation.....	81
6.4. Reproduction of human postural sway.....	82
6.5. CFD analysis results and discussion	85
6.5.1. Airflow and temperature distribution.....	85
6.5.2. Convective heat transfer coefficient on the CSP surface	90
6.5.3. Breathing zone	91
6.6. Limitation.....	97
6.7. Conclusion	98
REFERENCES.....	99

CHAPTER 7: Summary and Future Work

7.1. Summary of the dissertation	101
7.2. Future work.....	104
PUBLISHED ARTICLES RELATED TO THIS DISSERTATION.....	106

This page intentionally left blank

CHAPTER 1

Introduction

1.1. Research background and purpose

Individuals spend majority of their time indoors, where the quality of the indoor environment critically affects their health, comfort, and productivity [1-4]. With this, there is an ongoing demand for improved indoor conditions to better meet the needs of occupants. In addition, physiological functions of occupants and aspects of indoor environments interact with each other, significantly impacts the energy efficiency of buildings. Thus, analyzing these interactions to maintain appropriate indoor environmental quality is crucial for promoting a healthier, more comfortable, and energy-efficient indoor experience [5].

In response, numerous studies have been conducted to assess and improve indoor environmental quality (IEQ). The representation of the human body in indoor spaces has emerged as a crucial factor for accurately gauging the interactions between occupants and their environment. Many experimental studies have targeted the indoor environment and the occupants' immediate surrounding using thermal manikins or experimental subjects [6-12]. However, experimental measurements face several constraints including time, cost, scope, difficulty in reproducing complex phenomena, and ethical issues concerning the experimental subjects. For this reason, it is nearly impossible to conduct experiments with human subjects during the design process. Considering these issues, several comprehensive IEQ assessment methodologies have been proposed using numerical analysis, including Computational Fluid Dynamics (CFD) [13-20].

Along with this, computer-simulated persons (CSPs), which are entities capable of addressing specific human body characteristics as well as providing detailed geometries of the human body,

have been proposed and developed as a 'digital twin' for CFD analysis. Initially, simple shaped CSPs were used [21-24], but research has shown that the accuracy of human body representation significantly influences the precision of predictions and interpretations in simulations [25,26]. Consequently, there is a continuous demand for more detailed CSPs that closely mimic the real human body or thermal manikins, reflecting the continuous need for improved accuracy [27-29].

Along with the accurate CSP shape, it is crucial for the clothing model to precisely reflect the heat and moisture transfer characteristics of each body part. For this purpose, various thermoregulation models have been integrated with mathematical clothing models to represent clothing conditions. However, there are difficulties in measuring and quantifying convective heat transfer and ventilation within clothing, often leading to a simplification of heat transfer between clothing and skin. Experiments and numerical methods have shown that the air layer between clothing and skin plays a significant role [30-32]. Previous research has improved the CSP model originally developed by Kang et al.[33], with enhancements by Josh et al.[34], considering the ventilation layer between clothing and skin and validating it against existing experimental literature. However, for conducting visual and detailed analyses such as temperature and airflow distribution between the skin and clothing, geometrically accurate clothing models are essential.

In addition to the geometric model of the CSP, the focus has been on accurately representing the fundamental physiological functions of the human body, such as thermoregulation and breathing, which play a crucial role in evaluating indoor air quality and the thermal comfort [35-46]. Thermoregulation models are used to maintain the physiological temperature range of the human body or thermal manikin in response to the surrounding environment. These models may consider the human body as a single segment or divide it into multiple segments and may be classified as having one-, two-, or multi-node models [47,48]. The two-node model is widely

used due to its simplicity and reasonable accuracy [49]. However, it simplifies the complex systems of the body into just the 'core' and 'skin,' potentially causing errors during temperature changes by lumping multiple data points together for analysis. Additionally, it's mostly applicable to uniform environmental conditions with moderate physical activity. To overcome these limitations, a multi-node model has been proposed, which divides the body into multiple nodes representing different tissues, allowing for a more detailed temperature distribution. This multi-node model has been suggested to obtain a more detailed temperature distribution by dividing the human body into multiple nodes representing each tissue [50, 51].

Also, replicating human respiratory patterns is indeed critical for assessing indoor environments and predicting human health risks. Especially after global pandemics such as SARS-CoV-2, the identification of breathing zones (BZs) has become an important topic in the public health field for tracking airborne transmission through respiratory aerosol or for guidelines for proper social distancing [52, 53]. Although there is no clear definition of a BZ, it conventionally refers to the area around the occupant's nose and mouth from which they inhale, encompassing the space where airborne particles and contaminants are most likely to be breathed in. It has a significant impact on the assessment of pollutant transmission and exposure [54-58].

On the other hand, several studies have investigated the impact of physical activities like walking or running on indoor environmental quality (IEQ) and the microclimate around the human body. Such activities modify the microclimate surrounding the body, significantly influencing heat transfer and contaminant concentrations within the breathing zone [59-61]. Besides these active movements, the human body also experiences continuous micromovements while standing or sitting [62]. These movements, often manifested as postural sway, caused by variations in torque at the ankles, knees, and hip joints, and are regulated by neural control

pathways. These micro-movements are small but persistent, indicating they could have a lasting impact on indoor environments, yet have been largely neglected in IEQ research. Yoo et al. assumed the subject was a young, healthy adult under typical indoor environmental conditions and confirmed through the reproduction of micro-movements that the effect on convective heat and mass transfer was negligible, but suggested the impact might vary by the subject's conditions [62]. Despite findings that physical activities significantly impact contaminant concentrations in occupant's BZ, the mentioned studies related to the definition of BZ did not consider the human body's micro-movements. Therefore, there is a need to evaluate the effects of micro-movements in defining the BZs.

Considering this background, this study developed a digital twin of an ANDI thermal manikin, in which thermal function and skin temperature are regulated by a multi-node thermoregulation model, in both clothed and naked states. Geometrically heterogeneous clothing models were applied for the representation of clothing. Additionally, this study introduces a validation procedure for the prediction accuracy of the numerical thermoregulation model applied to the CSP using the CFD, comparing chamber experimental results derived from a thermal manikin. Additionally, the developed CSP was integrated with a respiratory system to replicate human breathing. We investigated the impact of micro-movements on the microclimate around the human body and the size/ shape of the BZs. To qualitatively and quantitatively identify the BZs, we utilized the Scale for Ventilation Efficiency (SVE)4 and 5, which estimate ventilation efficiency. Furthermore, a transient breathing model, including both inhalation and exhalation, was employed for a more realistic identification of the BZs.

1.2. Composition of dissertation

The structure of this dissertation, which aims to develop an improved and comprehensive CSP for evaluating indoor environments and assessing human risk using CFD analysis, is as follows:

Chapter 2 introduces the governing equations used in the numerical analyses conducted in this research, including the basic equations of flow fields, turbulence models, heat transfer and radiation models, and scalar transport equations.

Chapter 3 discusses the process of establishing the geometry of the CSP based on 3D scanning data of the target model which is the ANDI thermal manikin. This chapter illustrates the process of replicating the actual shape of clothing and integrating it with the CSP. Additionally, this introduces the Fiala multi-node thermoregulation model that controls the thermal functions and skin temperature of the ANDI thermal manikin in response to the surrounding environment, and also addresses the application of this model to the CSP integrated with CFD.

In Chapter 4, to validate the prediction accuracy of numerical thermoregulatory analysis coupled with computational fluid dynamics (CFD) simulation, the analysis results were compared with the experimental results using ANDI thermal manikin.

Chapter 5 explores the integration of a nasal cavity model into the developed CSP to replicate the breathing function of human. It introduces a transient breathing cycle model that simulates variable breathing flow rates over time, facilitating dynamic numerical analysis. Additionally,

this chapter discusses the use of SVE 4 and SVE 5 for analyzing the Breathing Zone (BZ), which is essential for evaluating contaminant transfer and assessing human exposure.

Chapter 6 discusses micro-movements resulting from posture sway, another characteristic of human physiology, and introduces the single-mass inverted pendulum model (IPM) developed to simulate these movements. Additionally, the chapter presents the results of CFD analysis that explores how these micro-movements influence the formation of the CSP's BZ and affect the general indoor climate.

In Chapter 7, the outcomes and achievements obtained from each chapter are summarized and described. Additionally, this chapter outlines the future tasks planned for this research.

The diagram structure of this dissertation is demonstrated in Fig 1.1, which graphically describes the structure.

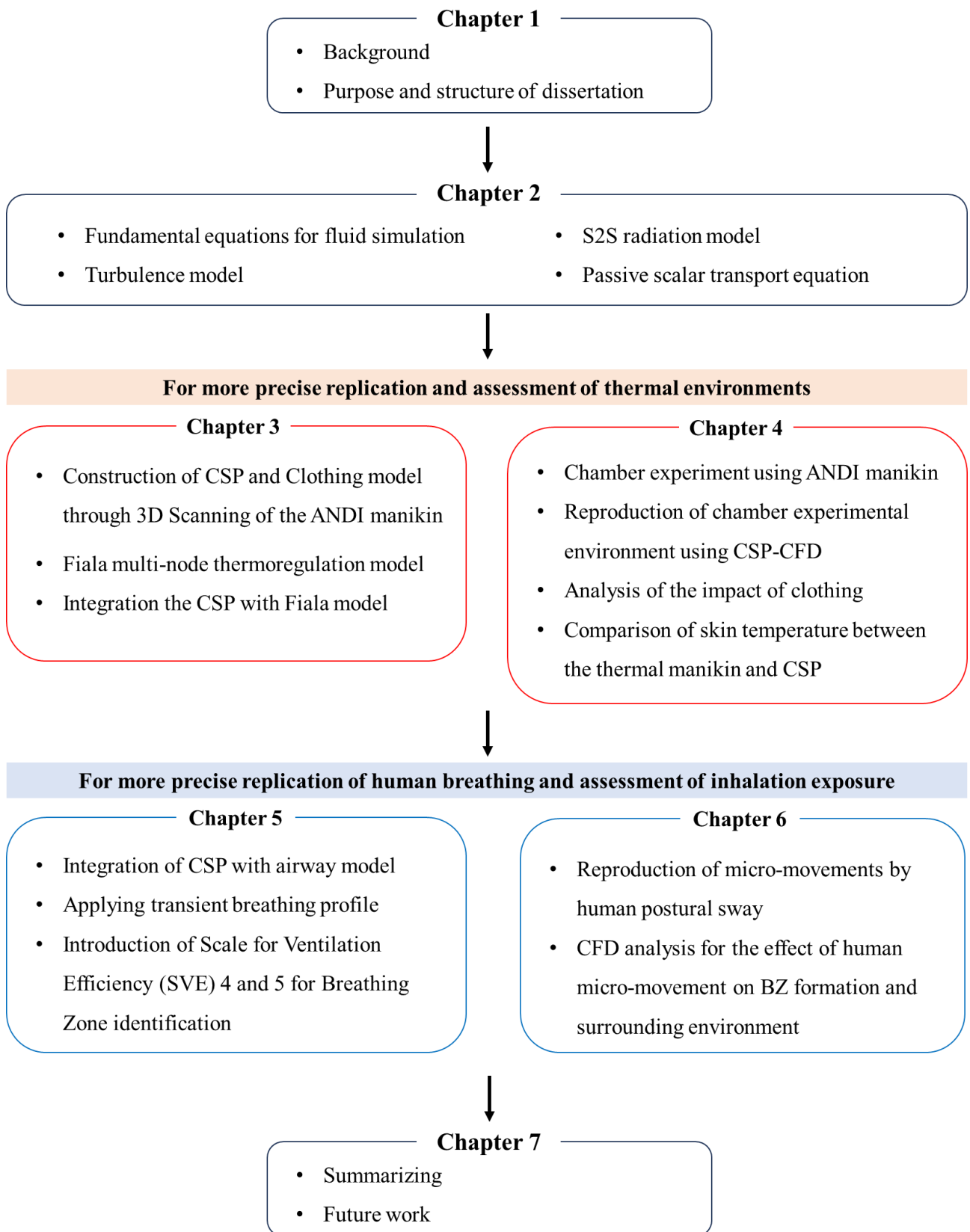


Figure 1.1. Diagram of dissertation structure.

REFERENCES

- [1] I. Mujan, A.S. Anđelković, V. Munćan, M. Kljajić, D. Ružić, Influence of indoor environmental quality on human health and productivity-A review, *J. Clean. Prod.* 217 (2019) 646–657.
- [2] N.E. Klepeis, W.C. Nelson, W.R. Ott, J.P. Robinson, A.M. Tsang, P. Switzer, J.V. Behar, S.C. Hern, W.H. Engelmann, The National Human Activity Pattern Survey (NHAPS): a resource for assessing exposure to environmental pollutants, *J. Expo. Sci. Environ. Epidemiol.* 11 (2001) 231–252.
- [3] X. Shan, A.N. Melina, E.-H. Yang, Impact of indoor environmental quality on students' wellbeing and performance in educational building through life cycle costing perspective, *J. Clean. Prod.* 204 (2018) 298–309.
- [4] D.P. Wyon, The effects of indoor air quality on performance and productivity, *Indoor Air.* 14 (2004) 92–101.
- [5] T. Catalina, V. Iordache, IEQ assessment on schools in the design stage, *Build. Environ.* 49 (2012) 129–140.
- [6] K. Ito, S. Murakami, Cost-effectiveness analysis of improved indoor temperature and ventilation conditions in school buildings, *J. Asian Archit. Build. Eng.* 9 (2010) 523–529.
- [7] K. Suehiro, N. Suehiro, Y. Masuda, K. Ito, K. Ito, Folded roof house—Design solution to optimize environmental conditions with generic equipment and techniques, *Jpn. Archit. Rev.* 1 (2018) 18–23.
- [8] M. Frontczak, P. Wargocki, Literature survey on how different factors influence human comfort in indoor environments, *Build. Environ.* 46 (2011) 922–937.
- [9] S. Tanabe, E.A. Arens, F. Bauman, H. Zhang, T. Madsen, Evaluating thermal environments by using a thermal manikin with controlled skin surface temperature, *ASHRAE Trans.* (1994) 3739, 39.
- [10] I. Reda, R.N. AbdelMessih, M. Steit, E.M. Mina, Experimental assessment of thermal comfort and indoor air quality in worship places: The influence of occupancy level and period, *Int. J. Therm. Sci.* 179 (2022) 107686.
- [11] P.V. Nielsen, The importance of a thermal manikin as source and obstacle in full-scale experiments, *Proceedings of the 3rd International Meeting on Thermal Manikin Testing 3IMM*, Stockholm, Sweden. (1999)

- [12] M. Veselý, P. Molenaar, M. Vos, R. Li, W. Zeiler, Personalized heating—Comparison of heaters and control modes, *Build. Environ.* 112 (2017) 223–232.
- [13] P.V. Nielsen, Specification of a two-dimensional test case, Aalborg University, IEX Annex 20: Air Flow Patterns within Buildings. (1990).
- [14] P.V. Nielsen, Fifty years of CFD for room air distribution, *Build. Environ.* 91 (2015) 78–90.
- [15] K. Ito, K. Inthavong, T. Kurabuchi, T. Ueda, T. Endo, T. Omori, H. Ono, S. Kato, K. Sakai, Y. Suwa, CFD benchmark tests for indoor environmental problems: Part 1 isothermal/non-isothermal flow in 2D and 3D room model, *Int. J. Archit. Eng. Technol.* 2 (2015) 1–22.
- [16] K. Ito, K. Inthavong, T. Kurabuchi, T. Ueda, T. Endo, T. Omori, H. Ono, S. Kato, K. Sakai, Y. Suwa, CFD benchmark tests for indoor environmental problems: Part 4 air-conditioning airflows, residential kitchen airflows and fire-induced flow, *Int. J. Archit. Eng. Technol.* 2 (2015) 76–102.
- [17] K. Ito, K. Inthavong, T. Kurabuchi, T. Ueda, T. Endo, T. Omori, H. Ono, S. Kato, K. Sakai, Y. Suwa, CFD benchmark tests for indoor environmental problems: Part 2 cross-ventilation airflows and floor heating systems, *Int. J. Archit. Eng. Technol.* 2 (2015) 23–49.
- [18] D.N. Sørensen, P.V. Nielsen, Quality control of computational fluid dynamics in indoor environments, *Indoor Air.* 13 (2003) 2–17.
- [19] T.G. Foat, S.T. Parker, I.P. Castro, Z.T. Xie, Numerical investigation into the structure of scalar plumes in a simple room, *J. Wind Eng. Ind. Aerodyn.* 175 (2018) 252–263.
- [20] T. Zhang, Y. Zhang, A. Li, Y. Gao, Y. Rao, Q. Zhao, Study on the kinetic characteristics of indoor air pollutants removal by ventilation, *Build. Environ.* 207 (2022) 108535.
- [21] H. Xing, A. Hatton, H.B. Awbi, A study of the air quality in the breathing zone in a room with displacement ventilation, *Build. Environ.* 36 (2001) 809–820.
- [22] H. Brohus, K.D. Balling, D. Jeppesen, Influence of movements on contaminant transport in an operating room., *Indoor Air.* 16 (2006) 356–372.
- [23] S. Murakami, S. Kato, J. Zeng, Flow and temperature fields around human body with various room air distribution, Part 1-CFD study on computational thermal manikin, *ASHRAE Trans.* 103 (1997) 3–15.
- [24] S. Murakami, S. Kato, J. Zeng, Combined simulation of airflow, radiation and moisture transport for heat release from a human body, *Build. Environ.* 35 (2000) 489–500.

- [25] C. Topp, P.V. Nielsen, D. Sorensen, Application of computer simulated persons in indoor environmental modeling/Discussion, *ASHRAE Trans.* 108 (2002) 1084–1084.
- [26] D.N. Sørensen, L.K. Voigt, Modelling flow and heat transfer around a seated human body by computational fluid dynamics, *Build. Environ.* 38 (2003) 753–762.
- [27] N.P. Gao, H. Zhang, J.L. Niu, Investigating indoor air quality and thermal comfort using a numerical thermal manikin, *Indoor Built Environ.* 16 (2007) 7–17.
- [28] P.V. Nielsen, S. Murakami, S. Kato, C. Topp, J.-H. Yang, Benchmark tests for a computer simulated person, Aalborg University, *Indoor Environmental Engineering*. (2003).
- [29] Y. Tao, W. Yang, K. Inthavong, J. Tu, Indoor particle inhalability of a stationary and moving manikin, *Build. Environ.* 169 (2020) 106545–106545.
- [30] B. Choudhary, F. Wang, Y. Ke, J. Yang, Development and experimental validation of a 3D numerical model based on CFD of the human torso wearing air ventilation clothing, *Int. J. Heat Mass Transf.* 147 (2020) 118973.
- [31] Y. Ke, J. Li, G. Havenith, An improved experimental method for local clothing ventilation measurement, *Int. J. Ind. Ergon.* 44 (2014) 75–81.
- [32] H.H. Pennes, Analysis of tissue and arterial blood temperatures in the resting human forearm, *J. Appl. Physiol.* 1 (1948) 93–122.
- [33] Z. Kang, F. Wang, An advanced three-dimensional thermoregulation model of the human body: development and validation, *Int. Commun. Heat Mass Transf.* 107 (2019) 34–43.
- [34] A. Joshi, F. Wang, Z. Kang, B. Yang, D. Zhao, A three-dimensional thermoregulatory model for predicting human thermophysiological responses in various thermal environments, *Build. Environ.* 207 (2022) 108506.
- [35] N. Mahyuddin, H.B. Awbi, E.A. Essah, Computational fluid dynamics modelling of the air movement in an environmental test chamber with a respiring manikin, *J. Build. Perform. Simul.* 8 (2015) 359–374.
- [36] S. Yoo, K. Ito, Multi-stage optimization of local environmental quality by comprehensive computer simulated person as a sensor for HVAC control, *Adv Build. Energy Res.* 14 (2020) 171–188.
- [37] K. Ito, Toward the development of an in silico human model for indoor environmental design, *Proc. Jpn. Acad. B.* 92 (2016) 185–203.

- [38] K. Kuga, P. Wargocki, K. Ito, Breathing zone and exhaled air re-inhalation rate under transient conditions assessed with a computer-simulated person, *Indoor Air*. 32 (2022) e13003.
- [39] K. Kuga, K. Ito, W. Chen, P. Wang, K. Kumagai, A numerical investigation of the potential effects of e-cigarette smoking on local tissue dosimetry and the deterioration of indoor air quality, *Indoor Air*. 30 (2020) 1018–1038.
- [40] K. Murota, Y. Kang, S. Hyodo, S.-J. Yoo, K. Takenouchi, S. Tanabe, K. Ito, Hygro-thermo-chemical transfer analysis of clothing microclimate using three-dimensional digital clothing model and computer-simulated person, *Indoor Built Environ*. 31 (2022) 1493–1510.
- [41] S. Yoo, K. Ito, Numerical prediction of tissue dosimetry in respiratory tract using computer simulated person integrated with physiologically based pharmacokinetic–computational fluid dynamics hybrid analysis, *Indoor Built Environ*. 27 (2018) 877–889.
- [42] S. Yoo, K. Ito, Assessment of transient inhalation exposure using in silico human model integrated with PBPK-CFD hybrid analysis, *Sustain Cities Soc*. 40 (2018) 317–325.
- [43] S. Yoo, K. Ito, Validation, verification, and quality control of computational fluid dynamics analysis for indoor environments using a computer-simulated person with respiratory tract, *Jpn. Archit. Rev*. 5 (2022) 714–727.
- [44] K. Ito, K. Inthavong, T. Kurabuchi, T. Ueda, T. Endo, T. Omori, H. Ono, S. Kato, K. Sakai, Y. Suwa, CFD benchmark tests for indoor environmental problems: Part 3 numerical thermal manikins, *Int. J. Archit. Eng. Technol*. 2 (2015) 50–75.
- [45] W. Liu, T.T. Zhang, D. Lai, Inverse design of a thermally comfortable indoor environment with a coupled CFD and multi-segment human thermoregulation model, *Build. Environ*. 227 (2023) 109769.
- [46] H. Park, S. Yoo, S. Eisaku, H. Hiroshi, K. Ito, Numerical Thermal Comfort Analysis Using Combined Computer-Simulated Person with Clothing and Multi-Node Thermoregulation Model, in: *EDP Sciences*, 396 (2023) 01019.
- [47] L. Yi, L. Fengzhi, L. Yingxi, L. Zhongxuan, An integrated model for simulating interactive thermal processes in human-clothing system, *J. Therm. Biol*. 29 (2004) 567–575.
- [48] K. Katić, R. Li, W. Zeiler, Thermophysiological models and their applications: A review, *Build. Environ*. 106 (2016) 286–300.
- [49] A.P. Gagge, An effective temperature scale based on a simple model of human physiological regulatory response, *ASHRAE Trans*. 77 (1971) 247–262.

-
- [50] A. Munir, S. Takada, T. Matsushita, Re-evaluation of Stolwijk's 25-node human thermal model under thermal-transient conditions: Prediction of skin temperature in low-activity conditions, *Building and Environment*. 44 (2009) 1777–1787.
 - [51] D. Fiala, K.J. Lomas, M. Stohrer, Computer prediction of human thermoregulatory and temperature responses to a wide range of environmental conditions, *International Journal of Biometeorology*. 45 (2001) 143–159.
 - [52] J. Huang, P. Jones, A. Zhang, S.S. Hou, J. Hang, J.D. Spengler, Outdoor Airborne Transmission of Coronavirus Among Apartments in High-Density Cities, *Front. Built Environ*. 7 (2021) 666923.
 - [53] I. Abouelhamd, K. Kuga, S.-J. Yoo, K. Ito, Identification of probabilistic size of breathing zone during single inhalation phase in semi-outdoor environmental scenarios, *Building and Environment* 243 (2023) 110672.
 - [54] H. Brohus, Measurement of personal exposure using a breathing thermal manikin, (1997).
 - [55] C.E. de Normalisation, EN1540 workplace atmospheres–terminology, Comite Europeen de Normalisation, Brussels (1998).
 - [56] G. Lidén, J. Waher, Experimental investigation of the concept of a 'breathing zone' using a mannequin exposed to a point source of inertial/sedimenting particles emitted with momentum, *Annals of Occupational Hygiene* 54 (2010) 100–116
 - [57] F.R. Haselton, P.G. Sperandio, Convective exchange between the nose and the atmosphere, *Journal of Applied Physiology* 64 (1988) 2575–2581.
 - [58] Y. Shang, K. Inthavong, J. Tu, Detailed micro-particle deposition patterns in the human nasal cavity influenced by the breathing zone, *Computers & Fluids* 114 (2015) 141–150.
 - [59] B.A. Edge, E.G. Paterson, G.S. Settles, Computational study of the wake and contaminant transport of a walking human, (2005).
 - [60] Y. Tao, W. Yang, K. Inthavong, J. Tu, Indoor particle inhalability of a stationary and moving manikin, *Building and Environment* 169 (2020) 106545.
 - [61] J. Hang, Y. Li, R. Jin, The influence of human walking on the flow and airborne transmission in a six-bed isolation room: tracer gas simulation, *Building and Environment* 77 (2014) 119–134.
 - [62] S. Yoo, K. Ito, Validation, verification, and quality control of computational fluid dynamics analysis for indoor environments using a computer-simulated person with respiratory tract, *Japan Architectural Review*. 5 (2022) 714–727.
-

CHAPTER 2

Outline of Numerical Analysis

2.1. Introduction

For developing a CSP that enables the evaluation of indoor environments and human health risks, it is crucial to accurately predict the effects on the human body by generating the non-uniform flow field, temperature field, humidity field, and contaminant concentration field within the indoor space.

To achieve this, the study employs precise CFD (Computational Fluid Dynamics) based prediction methods and appropriate computational models to ensure accurate numerical analysis. This study utilized the commercial CFD software ANSYS FLUENT 2022 R2. CFD is founded upon the fundamental governing equations of fluid mechanics and thermodynamics such as the continuity equation, Navier-Stokes equations, and energy equation [1]. These equations enable the prediction of diverse fluid states and mass/heat transfer [2]. Using these CFD simulations, this research conducts four distinct types of numerical analyses:

- Flow Field Analysis - Analyzing the velocity distribution of indoor airflow.
- Temperature Field Analysis - Based on the results of the flow field analysis, this examination focuses on the distributions of indoor temperatures and CSP's skin temperatures through convective heat transfer.
- Radiative Heat Transfer Analysis - Utilizing the S2S (Surface to Surface) radiation model to analyze radiation.
- Moisture and Contaminant Transport Analysis - Employing passive scalar transport equations to study the movement of moisture and pollutants.

In this chapter, the fundamental fluid dynamics equations necessary for calculating the flow fields within the target model are discussed, along with the transport equations for thermal energy, moisture, and contaminants. Additionally, the turbulence models used in this research, the radiation models employed for the coupling of radiative heat transfer phenomena, and the scalar transport equations intended for predicting moisture and pollutants by assuming them as passive scalars are discussed.

2.1. Numerical analysis of flow field, temperature field, and humidity field

2.2.1. *Fundamental equations for fluid simulation*

Incompressible fluids are those whose density remains constant or does not change due to compression. While all fluids have some degree of compressibility, in many fluid dynamics problems, the changes in fluid density are so minor that they can be ignored, allowing for simplification of calculations by considering the fluid to be incompressible. The movement of fluids is defined by the continuity equation and the Navier-Stokes equations in the case of incompressible fluids.

Continuity equation:

$$\frac{\partial U_i}{\partial x_i} = 0 \quad (2.1)$$

Navier-Stokes equation:

$$\frac{\partial U_i}{\partial t} + \frac{\partial U_i U_j}{\partial x_j} = -\frac{1}{\rho} \cdot \frac{\partial P}{\partial x_i} + \frac{\partial}{\partial x_j} \left(\nu \frac{\partial U_i}{\partial x_j} \right) + f_i \quad (2.2)$$

This continuity equation expresses the principle of mass conservation within a fluid, assuming the fluid is incompressible. Here, U_i (m/s) represents the components of instantaneous fluid

velocity (u, v, w) , ρ (kg/m^3) is the density of the fluid, P ($\text{kg}/(\text{m}\cdot\text{s}^2)$) is the pressure, ν (m^2/s) is the kinematic viscosity, f_i is external body forces. The Navier-Stokes equation, shown in Equation (2.2), represents the conservation of momentum, derived from Newton's second law, and includes the buoyancy effect. Thermal energy transfer is modeled to predict temperature changes within the fluid, as shown in Equation (2.3). Here α (m^2/s) is the thermal diffusivity, ϵ , and S (K/s) is the heat source term. This equation represents the energy conservation and is derived from Fourier's law. Similarly, the movement phenomena of scalar quantities such as moisture and pollutants can be defined by the scalar transport Equation (2-4). ϕ (kg/kg') represents scalar quantities like moisture and pollutants, D is the diffusion coefficient for the substance. By coupling Equation (2-1) to (2-4), it is possible to grasp the phenomena of incompressible fluids and scalar transport within the fluid.

$$\frac{\partial \theta}{\partial t} + \frac{\partial(U_i \theta)}{\partial x_i} = \frac{\partial}{\partial x_i} \left(\alpha \frac{\partial \theta}{\partial x_i} \right) + S \quad (2.3)$$

$$\frac{\partial \phi}{\partial t} + \frac{\partial(U_i \phi)}{\partial x_i} = \frac{\partial}{\partial x_i} \left(D \frac{\partial \phi}{\partial x_i} \right) + S' \quad (2.4)$$

2.2.2. Nondimensionalization of the equation system

These fundamental equations undergo a nondimensionalization process, which facilitates comparison and computation among physical quantities with different units by integrating all relevant variables into a standardized form. When changing to nondimensionalizing form the Navier-Stokes Equation (2.2), representative length L_0 (e.g., size of the inflow boundary) and representative velocity U_0 (inflow velocity) are used. The resulting form is shown in Equation (2.5).

$$\frac{\partial U_i^*}{\partial t^*} + U_j^* \cdot \frac{\partial U_i^*}{\partial x_j^*} = -\frac{\partial P^*}{\partial x_i^*} + \frac{1}{\text{Re}} \cdot \frac{\partial^2 U_i^*}{\partial x_j^{*2}} - \frac{Gr}{\text{Re}^2} \quad (2.5)$$

$$\text{Re} = \frac{\rho U_0 L_0}{\mu} = \frac{U_0 L_0}{\nu} \quad (2.6)$$

$$\text{Gr} = \frac{g \beta \theta L_0^3}{\nu^2} \quad (2.7)$$

The subscripts * denotes dimensionless quantities, with $U_i^* = U_i / U_0$, $x_i^* = x_i / L_0$, $t^* = t / t_0 = t / (L_0 / U_0)$, and $P^* = P / P_0 = P / U_0^2$. The Reynolds number (Re) and the Grashof number (Gr) are important dimensionless numbers in fluid dynamics and heat transfer. The Reynolds number represents the ratio of inertial forces to viscous forces in a fluid, while the Grashof number considers the relative significance of buoyancy forces compared to viscous forces. In Equation (2.5), as the Re increases, the diffusion term due to viscosity decreases, while the convection term becomes relatively larger. The diffusion term helps to distribute momentum uniformly across the fluid space, whereas the convection term represents the movement of momentum along the flow of the fluid. As the convection term increases, the instability of the flow also increases. Moreover, if the Re is sufficiently high, the diffusion term becomes negligible compared to the convection term. In such cases, the fluid flow can transition to turbulent, and the Re at this point is referred to as the critical Re.

Grashof number is particularly significant in natural convection scenarios, where it represents the ratio of buoyancy forces to viscous forces. A higher Gr indicates that buoyancy forces dominate over viscous forces, leading to enhanced convection. In systems with high Gr, the fluid motion is significantly driven by buoyancy effects, which can lead to flow instability and turbulence if the Gr is sufficiently large. On the other hand, Equations (2.3) and (2.4) can also be nondimensionalized as follows:

$$\frac{\partial(\theta^* U_j^*)}{\partial x_j^*} + \frac{\partial \theta^*}{\partial t^*} = \frac{\partial}{\partial x_j^*} \left(\frac{1}{\text{Re}} \cdot \frac{1}{\text{Pr}} \cdot \frac{\partial \theta^*}{\partial x_j^*} \right) + S^* \quad (2.8)$$

$$\frac{\partial(\phi^* U_j^*)}{\partial x_j^*} + \frac{\partial \phi^*}{\partial t^*} = \frac{\partial}{\partial x_j^*} \left(\frac{1}{\text{Re}} \cdot \frac{1}{\text{Sc}} \cdot \frac{\partial \phi^*}{\partial x_j^*} \right) + S^* \quad (2.9)$$

$$\text{Pr} = \frac{\mu / \rho}{(\lambda / C_p \cdot \rho)} = \frac{C_p \mu}{\lambda} = \frac{\nu}{\alpha} \quad (2.10)$$

$$\text{Sc} = \frac{\mu}{\rho D} = \frac{\nu}{D} \quad (2.11)$$

Where $\theta^* = \theta / T_0$, and the Prandtl Number (Pr) represents the dimensionless number indicating the ratio of the thermal diffusivity $\alpha (= \lambda / C_p \cdot \rho)$ to the kinematic viscosity ν . $\phi^* = \phi / \phi_0$, and Schmidt Number (Sc) is a dimensionless quantity that represents the ratio of the mass diffusivity D to the molecular diffusivity ν .

2.2.3. Analysis of equation systems through averaging and ensemble averaging

In fluid dynamics, fundamental equations including the Navier-Stokes equations are essential for modeling complex flow phenomena. However, these equations contain nonlinear terms, making direct computation of solutions practically impossible. Such nonlinearity particularly complicates the computation when dealing with turbulent flows, which are irregular and highly variable. For instance, predicting turbulence phenomena requires analyzing physical variations corresponding to Kolmogorov's microscales. This demands analytical grid with more than 10^{13} , task that is extremely challenging with current computational technologies. The necessity for such many mesh points makes computation difficult due to the limitations of current computer performance. To address these challenges, averaging techniques are employed. Averaging

simplifies computations by taking the average of physical quantities over a specified time or space, thus aiding in understanding general phenomena. For example, Equation (2.12) represents ensemble averaging, which computes the average of the function $f_i(x, t)$ at location x and time t :

$$\overline{f_k(x_i, t)}_E = \lim_{N \rightarrow \infty} \left(\frac{1}{N} \sum_{k=1}^N f_i(x, t) \right) \quad (2.12)$$

This formula calculates the average of the fluid property $f_i(x, t)$ at a given time t and location x , deriving statistical characteristics that recur over various times and conditions. For example, it can be used to determine the average velocity field or pressure distribution. Equation (2.13) represents spatial filtering, which helps analyze the spatial variability of turbulent flows:

$$f(x_i)_F = \iiint_{-\infty}^{\infty} \left(\prod_{i=1}^3 G(x_i - x_i') \right) \cdot f(x_i') dx_i' \quad (2.13)$$

$G(x_i - x_i')$ is a filter function, which applies a one-dimensional filter function extended in three spatial dimensions. This method facilitates the averaging of physical quantities over extensive spaces, providing efficient analysis in modeling complex turbulent flows and optimizing the use of computational resources.

In the series of equations shown from Equation (2.1) to Equation (2.3), the following equations are obtained by separating the instantaneous values into mean and fluctuating quantities, and then performing ensemble averaging.

$$\frac{\partial \overline{U_i}}{\partial x_i} = 0 \quad (2.14)$$

$$\frac{\partial \overline{U_i}}{\partial t} + \frac{\partial \overline{U_i U_j}}{\partial x_j} = -\frac{1}{\rho} \cdot \frac{\partial \overline{P}}{\partial x_i} + \nu \frac{\partial}{\partial x_j} \left(\frac{\partial \overline{U_i}}{\partial x_j} + \frac{\partial \overline{U_j}}{\partial x_i} \right) - \frac{\partial}{\partial x_j} \overline{u_i' u_j'} - g_i \beta \overline{\theta} \quad (2.15)$$

$$\frac{\partial \overline{\theta}}{\partial t} + \frac{\partial (\overline{\theta U_j})}{\partial x_j} = \frac{\partial}{\partial x_j} \left(\left(\alpha + \frac{\nu_\theta}{\sigma_\theta} \right) \frac{\partial \overline{\theta}}{\partial x_j} - \overline{u_j' \theta'} \right) \quad (2.16)$$

$\bar{U}_i, \bar{P}, \bar{\theta}$ represent the average velocity, pressure, and temperature. U_i' and θ' denote the fluctuating values, which are the difference between the actual instantaneous values and their respective averages. Equation (2.15) is commonly referred to as the Reynolds equation. On the right-hand side of this equation, $\overline{u_i' u_j'}$ represents Reynolds stress, while $\overline{u_j' \theta'}$ in Equation (2.16) indicates temperature flux. Equations (2.14) to (2.16) describe the flow using only mean quantities, explicitly excluding Reynolds stress and temperature flux. To complete the system of equations, it is necessary to model these variables using mean quantities. Details on the gradient diffusion approximation for Reynolds stress and the eddy viscosity model will be provided in subsequent sections.

2.3. Turbulence model

In this section, we will explain the RANS (Reynolds Averaged Navier-Stokes) models, specifically the standard k- ϵ model and the SST (Shear Stress Transport) k- ω model.

2.3.1 Standard k- ϵ model

In Equation (2.15), it is necessary to close the system of equations by modeling the Reynolds Stress terms, which are the correlation terms of the velocity fluctuations, using average quantities or similar methods. For this purpose, here we use the eddy viscosity model, which assumes that Reynolds Stresses are proportionally related to the mean velocity gradients [3].

$$\overline{u_i' u_j'} = \frac{2}{3} k \delta_{ij} - \nu_t \left(\frac{\partial \bar{U}_i}{\partial x_j} + \frac{\partial \bar{U}_j}{\partial x_i} \right) \quad (2.17)$$

In Equation (2-15), Reynolds stress is calculated using the eddy viscosity coefficient as shown in the relations described by equations (2-17) and (2-18). The k - ε two-equation model defines the eddy viscosity coefficient using the turbulent energy k (m^2/s^2) and the dissipation rate of turbulent energy ε (m^2/s^3), by solving the transport equations for both values to compute k and ε . Here, δ_{ij} (-) represents the Kronecker delta which has a value of 0 when $i \neq j$, and equals 1 otherwise, and ν_t (m^2/s) is the eddy viscosity. This formulation is appropriate when the fluid compressibility cannot be ignored, and the changes in fluid density have a significant impact.

$$\nu_t = C_\mu \frac{k^2}{\varepsilon} \quad (2.18)$$

C_μ (-) is commonly set to 0.09, following the recommendation by Spalding and Launder [4, 5]. This constant is used in turbulence modeling to calculate the eddy viscosity coefficient, which is crucial for modeling turbulent flows. By subtracting Equation (2.15), the Reynolds equation, from Equation (2.2), the Navier-Stokes equation, a motion equation concerning the change in velocity is derived. This equation is further processed by multiplying it by the fluctuating component u_i' , taking an ensemble average, and applying the definition of turbulent energy $k = \frac{\overline{u_i' u_i'}}{2}$, the turbulence kinetic energy transport equation is organized into Equation (2-19).

$$\begin{aligned} & \frac{\partial k}{\partial t} + \overline{U_j} \frac{\partial k}{\partial x_j} + \overline{u_i' u_j'} \frac{\partial \overline{U_i}}{\partial x_j} + \frac{\partial}{\partial x_j} \overline{u_j' k} \\ &= -\frac{1}{\rho} \cdot \frac{\partial}{\partial x_j} \cdot \overline{p u_j'} + \nu \frac{\partial}{\partial x_j} \cdot \overline{u_i' \left(\frac{\partial u_i'}{\partial x_j} + \frac{\partial u_j'}{\partial x_i} \right)} - \nu \cdot \frac{\partial u_i'}{\partial x_j} \left(\overline{\left(\frac{\partial u_i'}{\partial x_j} + \frac{\partial u_j'}{\partial x_i} \right)} \right) \end{aligned} \quad (2.19)$$

This equation can be simplified to derive the modeled transport equation for k as follows. First, the second term on the right-hand side (transport of k due to molecular diffusion) and the

third term on the right-hand side (the rate at which turbulent energy k is converted to thermal energy due to kinematic viscosity = viscous dissipation rate) are rewritten as follows.

$$\begin{aligned}
 & \nu \frac{\partial}{\partial x_j} \cdot \overline{u_i' \left(\frac{\partial u_i'}{\partial x_j} + \frac{\partial u_j'}{\partial x_i} \right)} - \nu \cdot \overline{\frac{\partial u_i'}{\partial x_j} \left(\frac{\partial u_i'}{\partial x_j} + \frac{\partial u_j'}{\partial x_i} \right)} \\
 &= \nu \frac{\partial^2 k}{\partial x_j^2} + \nu \overline{\left(\frac{\partial u_i'}{\partial x_j} \cdot \frac{\partial u_j'}{\partial x_i} \right)} - \nu \overline{\left(\frac{\partial u_i'}{\partial x_j} \cdot \frac{\partial u_i'}{\partial x_j} \right)} - \nu \overline{\left(\frac{\partial u_i'}{\partial x_j} \cdot \frac{\partial u_j'}{\partial x_i} \right)} \\
 &= \nu \frac{\partial^2 k}{\partial x_j^2} - \nu \cdot \overline{\frac{\partial u_i'}{\partial x_j} \cdot \frac{\partial u_i'}{\partial x_j}}
 \end{aligned} \tag{2.20}$$

The second term on the right-hand side of Equation (2.20) is the definition of the dissipation rate of turbulent energy, ε . The dissipation rate of turbulent energy can be expressed using the turbulent velocity U_t and the length scale of the turbulence, as shown in the following equation.

$$\varepsilon = C \cdot \frac{U_t^3}{\ell} = C_\mu \frac{k^{3/2}}{\ell} \tag{2.21}$$

Therefore, C is used as a proportionality constant. If the length scale ℓ can be predetermined according to Reynolds numbers, there is no need to solve the transport equation for ε , and the system of equations defined by $\nu_t = k^{1/2} \cdot \ell$ can be closed. In this way, models that calculate only the transport equation for k are referred to as one-equation models. The third term on the left side (the generation of k by Reynolds Stress) is expressed using the eddy viscosity coefficient from Equation (2.17) as follows:

$$-\overline{u_i' u_j'} \frac{\partial \overline{U_i}}{\partial x_j} = \nu_t \left(\frac{\partial \overline{U_i}}{\partial x_j} + \frac{\partial \overline{U_j}}{\partial x_i} \right) \frac{\partial \overline{U_i}}{\partial x_j} - \frac{2}{3} k \delta_{ij} \frac{\partial \overline{U_i}}{\partial x_j} = \nu_t \left(\frac{\partial \overline{U_i}}{\partial x_j} + \frac{\partial \overline{U_j}}{\partial x_i} \right) \frac{\partial \overline{U_i}}{\partial x_j} \tag{2.22}$$

$$\frac{\partial}{\partial x_j} \overline{u_j' k'} + \frac{1}{\rho} \frac{\partial}{\partial x_j} \overline{u_j' p} = \frac{\partial}{\partial x_j} \left(-\frac{v_t}{\sigma_k} \cdot \frac{\partial k}{\partial x_j} \right) \quad (2.23)$$

$$\frac{\partial k}{\partial t} + \overline{U_j} \frac{\partial k}{\partial x_j} = \frac{\partial}{\partial x_j} \left(\left(v + \frac{v_t}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right) + P_k - \varepsilon \quad (2.24)$$

For the fourth term on the left side of Equation 2.19, which represents the turbulent diffusion of k , and the first term on the right side. σ_k is proposed as 1, and by organizing the above equation, the modeled transport equation for the turbulent energy k is derived. P_k represents the production of turbulent kinetic energy.

$$\begin{aligned} \frac{\partial \varepsilon}{\partial t} + \overline{U_j} \frac{\partial \varepsilon}{\partial x_j} = & -2\nu \frac{\partial^2 \overline{U_i}}{\partial x_k \partial x_j} \overline{u_j' \frac{\partial u_i'}{\partial x_k}} - 2\nu \frac{\partial \overline{U_i}}{\partial x_j} \left\{ \overline{\frac{\partial u_i'}{\partial x_k} \cdot \frac{\partial u_j'}{\partial x_k}} + \overline{\frac{\partial u_i'}{\partial x_k} \cdot \frac{\partial u_k'}{\partial x_j}} \right\} \\ & - 2\nu \overline{\frac{\partial u_i'}{\partial x_k} \frac{\partial u_j'}{\partial x_k} \frac{\partial u_i'}{\partial x_j}} - \frac{\partial}{\partial x_j} \left(\overline{\nu u_j' \frac{\partial u_i'}{\partial x_k} \frac{\partial u_i'}{\partial x_k}} \right) - \frac{2\nu}{\rho} \frac{\partial}{\partial x_j} \left(\overline{\frac{\partial u_i'}{\partial x_k} \frac{\partial p}{\partial x_i}} \right) + \nu \frac{\partial^2}{\partial x_j^2} \varepsilon - 2 \left(\nu \overline{\frac{\partial^2 u_i'}{\partial x_k \partial x_j}} \right)^2 \quad (2.25) \\ \varepsilon' = & \nu \overline{\frac{\partial u_i'}{\partial x_j} \cdot \frac{\partial u_i'}{\partial x_j}} \end{aligned}$$

To derive the transport equation for the turbulent energy dissipation rate ε , subtraction of the Reynolds Equation (2.15) from the Navier-Stokes Equation (2.2) was done, which provides the equation of motion for the fluctuations in velocity. Then, after differentiating the result with respect to x_k and multiplying each term by $2\nu \left(\frac{\partial u_i}{\partial x_k} \right)$, and then taking the ensemble average leads to the Equation (2.25).

In modeling the terms of Equation (2.25), approximations are applied to effectively represent the dynamics. As turbulence diffusion predominantly originates from velocity fluctuations with significant energy, it is appropriate to use k and ε to calculate the scales of turbulence diffusion. This approach forms the foundational premise of the k - ε model. The scales of turbulence (velocity

scale $L = k^{\frac{3}{2}}$, time scale $t = k / \varepsilon$, and length scale $u = k^{\frac{1}{2}}$ are defined accordingly. Utilizing these scales, the fourth and fifth terms on the right-hand side of Equation (2.25), which model the contributions of velocity and pressure fluctuations to the diffusion of ε , are developed. This modeling clarifies the mechanisms of energy transfer and diffusion in turbulent flows.

$$\begin{aligned}
 & -\frac{\partial}{\partial x_j} \left(\overline{\nu u_j' \frac{\partial u_i'}{\partial x_k} \frac{\partial u_i'}{\partial x_k}} \right) - \frac{2\nu}{\rho} \frac{\partial}{\partial x_j} \left(\overline{\frac{\partial u_i'}{\partial x_k} \frac{\partial p}{\partial x_i}} \right) = -\frac{\partial}{\partial x_j} \left(\overline{-u_j' \varepsilon} - \frac{2\nu}{\rho} \overline{\frac{\partial u_i'}{\partial x_k} \cdot \frac{\partial p}{\partial x_k}} \right) \\
 & = \frac{\partial}{\partial x_i} \left\{ C_\varepsilon \left(\frac{\ell^2}{t} \right) \frac{\partial \varepsilon}{\partial x_i} \right\} = \frac{\partial}{\partial x_i} \left\{ C_\varepsilon \frac{k^2}{\varepsilon} \cdot \frac{\partial \varepsilon}{\partial x_i} \right\} = \frac{\partial}{\partial x_i} \left\{ \frac{v_t}{\sigma_\varepsilon} \cdot \frac{\partial \varepsilon}{\partial x_i} \right\}
 \end{aligned} \tag{2.26}$$

In Equation (2.26), $C_\varepsilon \frac{k^2}{\varepsilon}$ represents the scalar quantity of isotropy regardless of the flow direction, hence it is called the isotropic diffusion model. In Equation (2.25), the production term of ε , where turbulent energy increases through interaction with the velocity gradients of the flow, is given by the first and second terms on the right-hand side. However, comparing the production term of the first term on the right-hand side with the dissipation term of the third term on the right-hand side, the dissipation term is generally considered to be much larger. Additionally, the production term of the second term on the right-hand side becomes 0 for $i \neq j$ due to the isotropic dissipation process, and similarly becomes 0 for $i=j$ according to the continuity Equation (2-1). Therefore, the production term of ε is neglected in the modeling process.

Furthermore, the dissipation of ε is indicated by the third and seventh terms on the right-hand side, but these dissipation terms are modeled macroscopically under the assumption that the production term of $k(P_k)$ and the dissipation term of $k(\varepsilon)$ balance in the transport equation of k , following the local equilibrium process.

$$\begin{aligned}
 & -2\nu \frac{\partial u'_i}{\partial x_k} \frac{\partial u'_j}{\partial x_k} \frac{\partial u'_i}{\partial x_j} - 2 \left(\nu \frac{\partial^2 u'_i}{\partial x_k \partial x_j} \right)^2 = \frac{\varepsilon}{k} (C_{\varepsilon 1} \cdot P_k - C_{\varepsilon 2} \cdot \varepsilon) \\
 & = C_{\varepsilon 1} \cdot \frac{\varepsilon}{k} \cdot P_k - C_{\varepsilon 2} \cdot \frac{\varepsilon^2}{k}
 \end{aligned} \tag{2.27}$$

By rearranging Equation (2-25), the transport equation for the turbulent energy dissipation rate can be defined as in Equation (2-28).

$$\frac{\partial \varepsilon}{\partial t} + \overline{U_j} \frac{\partial \varepsilon}{\partial x_j} = \frac{\partial}{\partial x_j} \left(\left(\nu + \frac{\nu_t}{\sigma_\varepsilon} \right) \frac{\partial \varepsilon}{\partial x_j} \right) + C_{\varepsilon 1} \cdot \frac{\varepsilon}{k} \cdot P_k - C_{\varepsilon 2} \frac{\varepsilon^2}{k} \tag{2.28}$$

Here, the values proposed by Spalding and Launder, $C_{\varepsilon 1}=1.44$, $C_{\varepsilon 2}=1.92$, and $\sigma_\varepsilon=1.3$, are applied.

2.3.2 SST k - ω model

While the k - ε model is suitable for analyzing free stream or atmospheric flows, it falls short in accurately predicting flows near wall regions. To address these limitations, this research has adopted the SST (Shear Stress Transport) k - ω model. Based on the baseline k - ω model and incorporating Wilcox's standard k - ω model for boundary layer analysis, this hybrid approach is used to model free stream flows. The SST model enables more precise predictions near walls and throughout free streams by accurately calculating the shear stress, thereby effectively capturing the effects of walls. The k - ω model consists of equations for turbulent kinetic energy k and specific turbulent dissipation rate ω (where $\omega=\varepsilon/k$). Each transport equation is described as follows [4].

$$\frac{\partial k}{\partial t} + \overline{U_j} \frac{\partial k}{\partial x_j} = \frac{\partial}{\partial x_j} \left(\left(\nu + \frac{\nu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right) + P'_k - \beta \cdot k \omega \tag{2.29}$$

$$\frac{\partial \omega}{\partial t} + \overline{U_j} \frac{\partial \omega}{\partial x_j} = \frac{\partial}{\partial x_j} \left(\left(\nu + \frac{\nu_t}{\sigma_k} \right) \frac{\partial \omega}{\partial x_j} \right) + \alpha P_\omega - \beta \omega^2 \quad (2.30)$$

$$P_k' = \min(\nu_t S^2, 10\beta^* k \omega) \quad (2.31)$$

$$P_\omega = \frac{P_k}{\min\left(\frac{\alpha^* k}{\omega}, \frac{\alpha_1 k}{SF_2}\right)} \quad (2.32)$$

In the k - ω model, similar to the k - ε model, there is a term P_k' representing the generation of turbulent energy. However, when calculating P_k' , a minimum function is used to ensure that P_k does not exceed the value $10\beta k \omega$. This is particularly useful at stagnation points where turbulent energy can increase excessively, preventing physically unrealistic energy generation. By limiting P_k' in this manner, the model's physical accuracy and numerical stability are enhanced.

In the SST k - ω model, the addition of the new cross-diffusion term D_ω to the transport equation for ω enhances the model's accuracy in simulating turbulence dynamics. This term strengthens the interaction between ω and k , enabling more realistic representation of turbulence energy transport and distribution, especially near wall regions. This improvement allows the model to capture the complex behaviors of turbulent flows with greater precision.

$$\frac{\partial \omega}{\partial t} + \overline{U_j} \frac{\partial \omega}{\partial x_j} = \frac{\partial}{\partial x_j} \left(\left(\nu + \frac{\nu_t}{\sigma_k} \right) \frac{\partial \omega}{\partial x_j} \right) + \alpha P_k' - \beta \omega^2 + D_\omega \quad (2.33)$$

$$D_\omega = 2(1 - F_1) \frac{\sigma_{\omega,2}}{\omega} \frac{\partial k}{\partial x_j} \frac{\partial \omega}{\partial x_j} \quad (2.34)$$

The SST k - ω model uses the function F_1 to switch between the k - ε and k - ω turbulence models based on the flow conditions near walls. When $F_1=0$, the model behaves like the k - ε model, which is advantageous in free-stream regions where the flow is less affected by wall proximity. This switch allows the model to exploit the strengths of the k - ε model, such as its robustness in

handling high Reynolds number flows away from surfaces. Conversely, when $F_1=1$, it fully utilizes the $k-\omega$ model, which is more accurate for near-wall turbulence modeling due to its sensitivity to low Reynolds numbers and its ability to handle the viscous sublayer.

$$F_1 = \tanh \left[\left[\min \left[\max \left(\frac{\sqrt{k}}{\beta \omega y}, \frac{500\nu}{y^2 \omega} \right), \frac{4\sigma_{\omega 2} k}{D_{k\omega} y^2} \right] \right]^4 \right] \quad (2.35)$$

$$D_{k\omega} = \max \left[2\rho\sigma_{\omega 2} \frac{1}{\omega} \frac{\partial k}{\partial x_i} \frac{\partial \omega}{\partial x_i}, 10^{-10} \right] \quad (2.36)$$

Here, y represents the distance to the nearest wall. The function F_2 is used specifically in the calculation of the turbulent viscosity coefficient ν_t . While F_1 is directly used in the transport equations for ω to switch between the $k-\epsilon$ and $k-\omega$ models, F_2 influences how the model calculates turbulence viscosity near the wall. This function adjusts the turbulence modeling to be more accurate in regions close to solid boundaries by affecting terms related to viscosity in the model equations. F_2 is particularly important for ensuring that the model behaves appropriately under near-wall conditions where the flow dynamics are significantly different from those in the free stream.

$$\nu_t = \rho \frac{k}{\omega} \min \left(\alpha^*, \frac{\alpha_1 \omega}{SF_2} \right) \quad (2.37)$$

$$F_2 = \tanh \left[\frac{2\sqrt{k}}{\beta^* \omega y}, \frac{500\nu}{\omega y^2} \right] \quad (2.38)$$

where, $\alpha^*=1$ and $\alpha_1=0.31$ in Menter's SST $k-\omega$ model, which is the coefficient describing the nonlinear transition of the flow regime from a laminar flow to a turbulent flow in which the relationship $\beta^*=0.09$ is satisfied. ($\sigma_{k1}=1.176$, $\sigma_{k2}=1$, $\sigma_{\omega 1}=2$, $\sigma_{\omega 2}=1.168$) [6, 7]

2.4. S2S (Surface to Surface) radiation model

The S2S (Surface-to-Surface) model is developed to precisely analyze radiative heat transfer between grey diffusive surfaces in enclosed environments, integrated into the commercial CFD software ANSYS FLUENT. This model calculates energy transfer between surfaces based on various geometric factors such as surface size, distance between them, and orientation angles. These factors are defined through the view factor, predicting the exchange of radiative energy based on surface composition and placement.

The S2S model employs a simplified assumption that ignores the effects of absorption, reflection, and scattering that can occur during radiation. This results in the model only considering direct radiative exchange between surfaces, simplifying complex radiative phenomena for fast and efficient analysis.

The model is based on the grey body assumption, where all surfaces are considered to have wavelength-independent emissivity and absorptivity, as per Kirchhoff's Law, implying that emissivity and absorptivity are equal ($\varepsilon = \alpha$), crucial for maintaining energy balance. It also assumes diffusive reflection, meaning radiation and absorption are independent of direction.

In this model, the total energy flux emanating from any surface depends on the radiant energy incident on that surface, divided into reflection, absorption, and transmission. The grey diffusive model assumes surfaces are opaque, hence transmission is neglected, and according to the conservation of energy, the sum of reflectivity and absorptivity equals 1 ($\rho + a = 1$). This implies that if the emissivity is ε , then the reflectivity is $1 - \varepsilon$.

As shown in Equation (2.39), the energy flux emitted from a surface, $q_{out,k}$, is the sum of the energy directly emitted by the surface due to its temperature and the energy reflected from other

surfaces. The total emitted energy from surface k is calculated using its emissivity ε_k and Stefan-Boltzmann constant σ as $\varepsilon_k \sigma T_k^4$. The reflected component is the result of incident energy from other surfaces, $q_{in,k}$, that is not absorbed.

$$q_{out,k} = \varepsilon_k \sigma T_k^4 + \rho_k q_{in,k} \quad (2.39)$$

The incident energy $q_{in,k}$, on a particular surface k is collective contribution from all other surfaces j in the system, modified by the view factor F_{jk} that quantify the fraction of energy leaving surface j that reaches surface k . This is mathematically expressed as:

$$A_k q_{in,k} = \sum_{j=1}^N A_j q_{out,j} F_{jk} \quad (2.40)$$

where A_j is the area of surface j , and $q_{out,j}$ is the energy flux leaving surface j .

Using the reciprocity relationship between view factors, $A_j F_{jk} = A_k F_{kj}$, ensures that the energy exchange between any two surfaces is consistent regardless of the direction of the analysis. This principle is fundamental in verifying the consistency of radiative heat exchange models. The formula for the outgoing energy flux $q_{out,k}$ from surface k is then given by:

$$q_{out,k} = \varepsilon_k \sigma T_k^4 + \rho_k \sum_{j=1}^N F_{kj} q_{out,j} \quad (2.41)$$

where ε_k is the emissivity, σ is the Stefan-Boltzmann constant, T_k is the temperature of surface k , ρ_k is the reflectivity, and F_{kj} is the view factor from surface k to surface j . This equation accounts for both the energy directly emitted by the surface and the energy reflected from other surfaces.

The term $\sum_{j=1}^N F_{kj} q_{out,j}$ represents the total radiative energy incident on surface k from all other surfaces, which contributes to the reflected component of $q_{out,k}$.

$$q_{out,k} = \varepsilon_k \sigma T_k^4 + \rho_k \sum_{j=1}^N F_{kj} q_{out,j} \quad (2.42)$$

To encapsulate the entire system's radiative transfers, we consider the energy radiated by each surface, J_k , and the emissive power E_k of each surface. The relationship between these can be expressed in a simplified matrix form as:

$$KJ = E \quad (2.43)$$

where J is the radiosity vector representing the total energy radiated by each surface, E is the vector of emissive powers, and K is a matrix derived from the view factors F_{kj} and the reflectivity ρ_k . The matrix equation $KJ=E$ essentially consolidates the individual radiative transfer equations for each surface into a system-wide equation that can be solved using numerical methods to find the radiosity J . This approach is very useful in complex geometries where direct calculation of each surface's radiative interactions would be cumbersome.

Additionally, the view factor F_{ij} between two finite surfaces i and j can be mathematically defined as:

$$F_{ij} = \frac{1}{A_i} \int_{A_i} \int_{A_j} \frac{\cos \theta_i \cos \theta_j \delta_{ij}}{\pi r^2} dA_i dA_j \quad (2.44)$$

where δ_{ij} is determined by the visibility between differential areas dA_i to dA_j ; $\delta_{ij}=1$ if dA_i is visible to dA_j and 0 otherwise. This formula calculates the fraction of energy leaving surface i that directly strikes surface j , considering geometric configurations and orientations of the surfaces relative to each other. By integrating these concepts, we gain a comprehensive

framework to analyze and predict the radiative heat transfer in systems composed of multiple interacting surfaces.

During the measurement times of S2S, the used surface area and the measurement temperatures are maintained constant. For each experimental condition, the heat transfer between S1 and S2 is calculated using a specific formula, where Δt represents the measurement time intervals, and the heat transferred during this period is summed and averaged. This method of measurement, which involves 20 measurements under each experimental condition, calculates the heat transfer coefficient using the average of all measurements, details of which are illustrated in the subsequent figures and tables. This approach allows us to efficiently manage computational resources by reducing the number of radiative surfaces through clustering. The clustering algorithm groups neighboring surfaces based on their proximity and similar characteristics, which simplifies the calculation by treating these groups as single units. The key equation used in the clustering process is:

$$T_{SC} = \left(\frac{\sum (A_f T_f^4)}{\sum A_f} \right)^{\frac{1}{4}} \quad (2.45)$$

This equation calculates the average temperature of a surface cluster, T_{SC} , where A_f and T_f represent the area and temperature of each face within the cluster, respectively. The use of the fourth power of the temperature reflects the non-linear nature of radiation and taking the fourth root aids in averaging these effects across the cluster.

Additionally, the smoothing process ensures that the results are reliable and consistent across different simulation scenarios. Smoothing is applied to the view factor matrix to enforce the reciprocity relationship and conservation of energy. This relationship is expressed by the equation:

$$A_i F_{ij} = A_j F_{ji} \quad (2.46)$$

where A_i and A_j are the areas of surfaces i and j , and F_{ij} is the view factor between these surfaces, indicating the proportion of radiation leaving surface i that strikes surface j . This ensures that the radiation exchange is symmetrical and conserves energy.

2.5. Passive scalar transport equation

In this study, we assume the moisture and tracer gas for predicting the BZ distribution as passive scalars. The humidity and tracer gas concentration fields were calculated using the scalar transport equation.

$$\frac{\partial \bar{\phi}}{\partial t} + \frac{\partial (\bar{\phi} \bar{U}_i)}{\partial x_i} = \frac{\partial}{\partial x_i} \left(\left(D + \frac{v_t}{\sigma_\phi} \right) \frac{\partial \bar{\phi}}{\partial x_i} \right) + S' \quad (2.47)$$

The ensemble averages are denoted by the overlines, $\phi(-)$ is the concentration of scalar at a grid point, U_i (m/s) represents the velocity (u, v, w components), and v_t (m²/s) is the turbulent viscosity. D_ϕ (m²/s) is the scalar diffusivity and $\sigma_\phi(-)$ is the turbulent Schmidt number.

REFERENCES

- [1] H.K. Versteeg, W. Malalasekera. An introduction to Computational Fluid Dynamics: The finite volume method. Pearson Education Limited. England; 2007. 503 p.
- [2] T. Wah-Yen, Y. Asako, N. Azwadi, C. Sidik, G. Rui-Zher. Governing Equations in Computational Fluid Dynamics: Derivations and A Recent Review. Progress in Energy and Environment. 2017;1 (June):1–19.
- [3] J. Boussinesq. Theorie Analytique de la Chaleur, 2:157-158, Gauthier-Villars, Paris., 1903
- [4] B. E. Launder, D.B. Spalding, The numerical computation of turbulent flows, in: Numerical Prediction of Flow, Heat Transfer, Turbulence and Combustion, Elsevier, 1983: pp. 96–116.
- [5] D. B. Spalding, Mathematical models of turbulent flames; a review, Combustion Science and Technology 13 (1976) 3–25.
- [6] F. R. Menter. Two-equation eddy-viscosity turbulence models for engineering applications. AIAA Journal. 1994;32(8):1598–605.
- [7] F. R. Menter, M. Kuntz, R. Langtry, Ten years of industrial experience with the SST turbulence model, Heat and Mass Transf. 4 (2003) 625–632.
- [8] M. F. Modest, S. Mazumder, Radiative heat transfer, Academic press, 2021.
- [9] M. E. Larsen, J.R. Howell, Least-Squares Smoothing of Direct-Exchange Areas in Zonal Analysis, Journal of Heat Transfer 108 (1986) 239–242.

Development of a CSP integrated with multi-node thermoregulation model**3.1. Introduction**

This research aims to develop a comprehensive indoor environment and human risk assessment method by integrating numerical human models and respiratory models. Therefore, Enhancing the accuracy of predicting heat and moisture transfer around the human body by incorporating a multi-node thermoregulation model is reasonable. In this study, an ANDI thermal manikin (Thermetrics), which regulates heat functions and skin temperature based on a multi-node thermal regulation model, was selected as the target model. This choice aims to validate the thermal prediction accuracy of the developed CSP by comparing it with actual chamber experiment results using the ANDI thermal manikin.

3.2. Overview of numerical human model

To evaluate the non-stationary and non-uniform environments formed indoors, numerous numerical human models (often referred to as Numerical Thermal Manikin, Virtual Manikin, or Computer Simulated Person) have been developed, accurately reproducing the human form. In this dissertation, the term "CSP" will be used to denote these models. In the early stages, relatively simple shapes, such as cylinders and cuboids, were used for human models in CFD analysis due to the limitations of computational resources [1,2]. Murakami et al. [3,4] developed a simplified non-cuboid CSP based on the real human body shape. However, because research findings have shown that the accuracy of the human body shape significantly affects the accuracy of interpretation and prediction around the human body [5,6], various CSPs that precisely reproduce

the real human body shape or thermal manikin have been proposed, and there is a continuous demand for improved CSPs [7-9]. Furthermore, to enhance the accuracy of indoor thermal environment evaluations, various human thermoregulation models proposed to date have been integrated into CSP. These models report methods and results that predict the heat balance with the indoor environment and the convective, radiative, and latent heat losses on the human skin surface.

3.3. Establishment process of CSP's geometry and clothing model

Geometric models play an essential role in CFD calculations. Therefore, the high-precision handheld 3D scanner was used to precisely establish the 3D geometry of CSP based on the ANDI thermal manikin. This scanner is notable for its Handheld HD Scan Mode which provides an increased scanning range and the capability to efficiently capture 3D models of objects ranging from medium to large sizes.



Figure 3.1. High-precision handheld 3D scanner (EinScan Pro 2X Plus; SHINING 3D, accuracy of $\pm 0.04\text{mm}$)

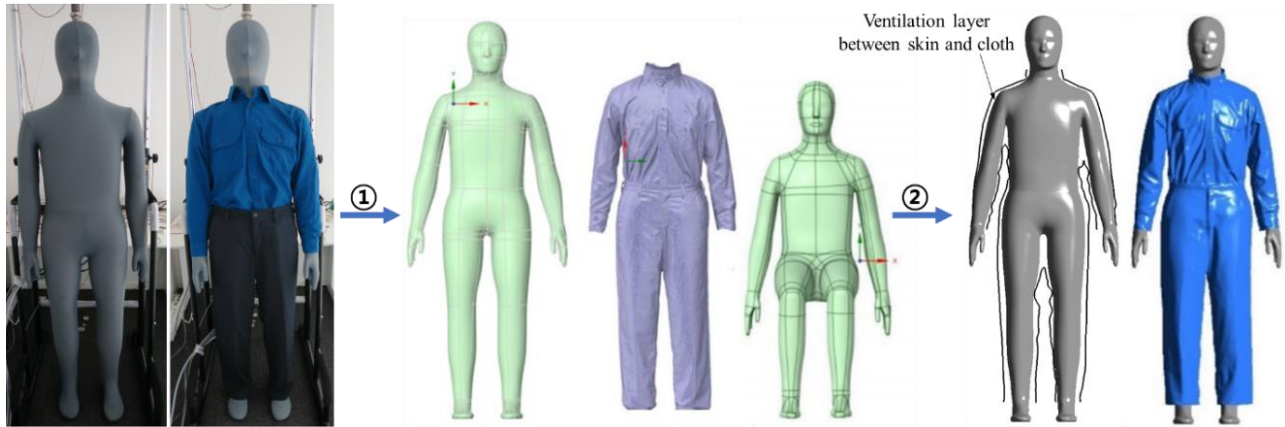


Figure 3.2. Process for creating CSP and clothing.

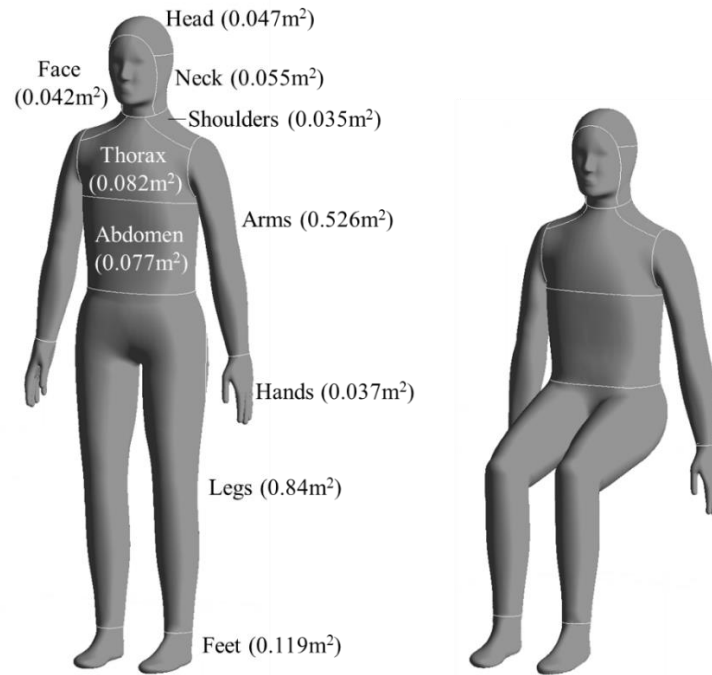


Figure 3.3. Segmentation of the CSP

After surface smoothing and segmentation, naked models (in the standing and seated postures) and a clothed model (in the standing posture) were created, as depicted Figure 3.2. Initially, the manikin was scanned separately in both its naked and clothed states to capture the distinct geometries of the manikin and the clothing. In the second step, the clothing model was integrated with the naked CSP to form a CSP with a heterogeneous clothing layer. The air gap between the manikin and clothing represented as the ventilation layer. Due to technical limitations

in grid generation within this study, the geometry of the skin-clothing contact area was adjusted. A minimum gap of 3mm was maintained across the CSP surface, with larger air gaps exceeding 5cm near areas such as the armpits or groin.

To ensure consistency in part segmentation between the thermal manikin and the CSP, the CSP's geometry adheres to the segmentation lines of the thermal manikin, utilizing the segmentation tool in Space-Claim software. Consequently, in line with the segment divisions of the Fiala model, the CSP is segmented into 15 parts as depicted in Figure 3.3: Head (0.047m^2), Face (0.042 m^2), Neck (0.055 m^2), Shoulders (0.035 m^2), Thorax (0.082 m^2), Arms (0.526 m^2), Hands (0.037 m^2), Abdomen (0.077 m^2), Legs (0.84 m^2), Feet (0.119 m^2).

3.4. Fiala thermoregulation model

3.4.1. Body construction

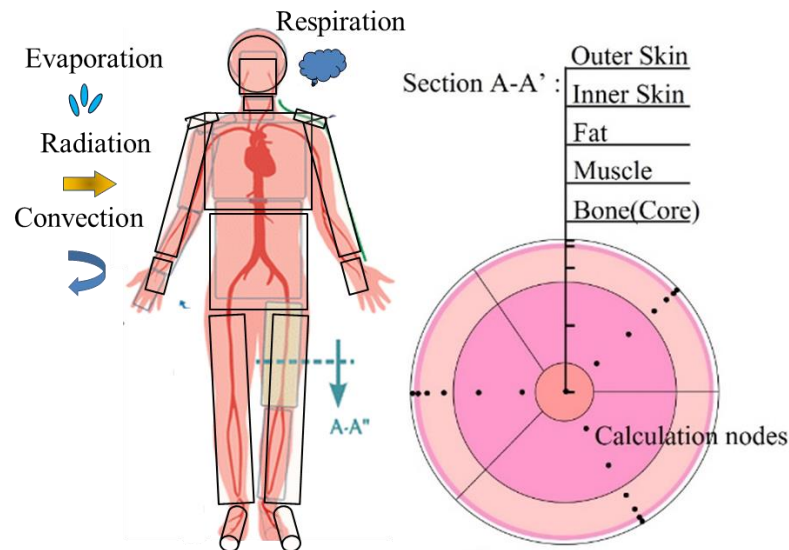


Figure 3.4. Body and tissue structure

The Fiala model, which regulates the thermal functions and skin temperature of the ANDI thermal manikin, assumes a standard adult male body. This model represents a subject with a

body weight of 73.5 kg, a body surface area of 1.86 m², and a basal metabolism rate of 87 W. As depicted as Figure 3.4, the human body in the model is segmented into 15 cylindrical or spherical elements: head, face, neck, thorax, abdomen, shoulders, arms, legs, hands, and feet. Each element consists of circular concentric tissue layers composed of seven different materials of tissue: brain, lung, bone, muscle, viscera, fat, and skin [10]. The heat transfer mechanisms in living tissues are described using the bioheat equation proposed by Pennes, as shown in Equation (3.1) [11]. The Fiala model is a multi-segment, multi-node thermal regulation model allows for the specification of the number of nodes within different tissue layers in the radial direction. Each computing nodes adheres to Pennes equation, which accounts for heat conduction, blood circulation, and metabolism.

$$k \left(\frac{\partial^2 T}{\partial r^2} + \frac{\omega}{r} \frac{\partial T}{\partial r} \right) + q_m + \rho_{bl} w_{bl} c_{bl} (T_{bla} - T) = \rho c \frac{\partial T}{\partial t} \quad (3.1)$$

3.4.2. Heat conduction

In Equation (3.1), the first term on the left-hand side represents heat conduction. Here, k (W/(m·K)) denotes the tissue conductivity, r (m) indicates the outer radius, and T (°C) refer to the tissue temperature. The variable ω (-) is a geometry factor, taking a value of 1 for polar coordinates and 2 for spherical coordinates, which is applicable to areas like the head. This differentiation is important because the temperature gradient from the manikin's skin surface to the human core varies. However, since heat primarily moves from the core outward, the transfer of heat between different parts of the manikin is generally considered negligible. Therefore, in this study, we focus solely on radial heat conduction, which is the heat moving outward from the center.

3.4.3. Metabolism

In Equation (3.1), the second term on the left-hand side, Q_m (W/m³), represents the total metabolic heat production occurring across various computing nodes in different segments. In Equation (3.2), the Q_m is divided into two parts: the basal metabolism $q_{m,bas,0}$, which maintains the normal physiological state of cells, and additional heat production, Δq_m . The additional heat primarily includes heat generated from activities such as exercise and shivering.

$$q_m = q_{m,bas,0} + \Delta q_m \quad (3.2)$$

$$\Delta q_m = \Delta q_{m,bas,0} + q_{m,sh} + q_{m,w} \quad (3.3)$$

$$\Delta q_{m,bas,0} = q_{m,bas,0} [2^{(T-T_0)/10} - 1] \quad (3.4)$$

$$q_{m,w} = \frac{\partial(a_{m,w}H)}{\partial V_{msc}} \quad (3.5)$$

In the Fiala model, the total basal metabolic rate at thermal neutrality, denoted as $M_{bas,0}$, is calculated to be 87W. This value aligns with the metabolic rate specified for a human model by ASHRAE Standard 55 and ISO 7730. $\Delta q_{m,bas,0}$ represents the deviation between the basal metabolic rate at thermal neutrality and the actual basal metabolic rate. $T - T_0$ represents the difference between the actual tissue temperature and the setpoint temperature. $q_{m,w}$ denotes additional metabolism due to work, while $q_{m,sh}$ represents additional metabolism by induced by shivering. $a_{m,w}$ is a coefficient that allocates the additional metabolic energy generated during exercise across different muscle volumes, thereby determining the distribution of extra metabolic

energy during physical activities. V_{msc} refers to the volume of muscle involved in an activity, quantifying the muscle mass that contributes to metabolic heat production during physical exertion.

$$H = M_{bas,0} \frac{act}{act_{bas}} (1 - \eta) - M_{bas,0} \quad (3.6)$$

$$\eta = 0.2 \cdot \tanh(b_1 \cdot act + b_0) \quad (3.7)$$

H represents the internal whole-body workload, indicating the total energy expenditure by muscles that is not converted into external mechanical work, but instead used to sustain muscle activity and heat the body. The actual metabolic rate is calculated using the activity level unit act (expressed in METs), which is compared to its baseline value of $act_{bas}=0.8$ METs.

In Equation (3.6), the term $1 - \eta$ represents the proportion of the total metabolic heat production that is utilized to warm the body, rather than being converted into external mechanical energy. Human mechanical efficiency is not constant and tend to increase with higher level of physical. At lower activity levels, the efficiency is notably low; however, as physical activity intensifies, efficiency also increases, potentially converting up to 20% of the body's metabolic energy into external work. To analyze this relationship, we employed regression analysis, utilizing measurements provided by Wyndham et al [12]. This analysis produced the coefficients b_1 and b_0 , which describe human work efficiency during activities like stair climbing or cycling ($b_1=0.39$ and $b_0=-0.6$ as reported in [10]). Specifically, at activity levels below 1.6 METs, efficiency is zero, indicating that energy is predominantly expended as heat. Between 1.6 and 5.0 METs, efficiency increases almost linearly with rising activity levels. At very high activity levels,

efficiency approaches a theoretical maximum of 20%.

3.4.4 Blood circulation

In Equation (3.1), the last term on the left-hand side represents blood circulation. The discussion on blood circulation in the model explores how the regulation of blood flow impacts body temperature control. The processes of heat storage and release within the bloodstream is vital for maintaining thermal equilibrium. According to the model, the human blood circulation system is composed of three primary components: 1) the central blood pool, 2) counter-current heat exchanges (CCX), and 3) pathways to individual tissue nodes. The entire body receives warm blood from the central pool via the main arteries. Before this blood is perfused into the tissues of limbs and shoulders, it is cooled through counter-current flows, during which heat is transferred to the adjacent veins.

In the process of blood circulation, arterial blood exchanges heat with surrounding tissues within the capillaries. During this interaction, blood either absorbs or releases heat, depending on the temperature differential. Following this, venous blood collects in the central vein, where it absorbs heat emitted from adjacent arteries, and then flows back to the central blood pool. Within this central blood pool, blood from different sources mixes, resulting in a new average temperature. This mechanism is crucial for the body's ability to maintain thermal homeostasis.

The incorporation of counter-current heat exchange (CCX) simulation into the model significantly enhances the realism of the heat exchange process. This addition allows for more accurate predictions of arterial blood temperatures ($T_{bl,a}$) across different parts of body, providing more accurate simulations than models that assume a uniform arterial blood temperature equal

to that of the central blood pool ($T_{bl,p}$). Adhering to the principle of mass conservation within blood vessels, any reduction in the arterial blood temperature ($T_{bl,p} - T_{bl,a}$) within a segment is counterbalanced by a corresponding increase in the venous blood temperature ($T_{bl,v,x} - T_{bl,v}$) in the same segment, as facilitated by the CCX mechanism:

$$\rho_{bl}c_{bl} \int w_{bl} dV (T_{bl,p} - T_{bl,a}) = \rho_{bl}c_{bl} \int w_{bl} dV (T_{bl,v,x} - T_{bl,v}) \quad (3.8)$$

This equation demonstrates the blood flow in the capillaries ($\int w_{bl} dV$) acts as the integrating factor for the thermal exchange, with $T_{bl,a}$ and $T_{bl,v,x}$ representing the temperatures of arterial and venous blood post-CCX, respectively. This method allows for the calculation of temperature shifts within the arterial and venous blood due to thermal exchanges. The following formula is then employed to calculate the net heat transfer between adjacent blood vessels (Q_x):

$$Q_x = h_x (T_{bl,a} - T_{bl,v}) \quad (3.9)$$

In this equation, h_x represents the counter-current heat exchange coefficient, and $T_{bl,v}$ indicates the venous blood temperature before the exchange. The arterial blood temperature, $T_{bl,a}$, can then be expressed using the following formula:

$$T_{bl,a} = \frac{\rho_{bl}c_{bl} \int w_{bl} dV \cdot T_{bl,p} + h_x \cdot T_{bl,v}}{\rho_{bl}c_{bl} \int w_{bl} dV + h_x} \quad (3.10)$$

The bioheat equation assumes that blood in capillaries reaches thermal equilibrium with the surrounding tissue. The $T_{bl,v}$ of a segment in Equation (3-9) is determined as follows:

$$T_{bl,v} = \frac{\int w_{bl} T dV}{\int w_{bl} dV} \quad (3.11)$$

The heat exchange coefficients (h_x) between adjacent arteries and veins are set to zero for

central body segments due to the absence of reliable data. For like the literature, these coefficients were estimated based on experimental data. For extremities such as legs, feet, arms, and hands, the h_x values were estimated from experimental data [10].

In non-neutral conditions or during physical activity, blood flow w_{bl} responds to variations in regional metabolic rates (q_m). Experimental evidence supports a linear relationship between w_{bl} and q_m . Consequently, the model does not directly handle blood perfusion rates; instead, it uses their energy equivalent, denoted as $\beta = \rho_{bl} c_{bl} w_{bl}$ ($W \cdot m^{-3} \cdot K^{-1}$). As the demand for oxygen rises, this is captured by computing the variation in the gain factor $\Delta\beta = \rho_{bl} c_{bl} \Delta w_{bl}$. This variation is modeled as a function of changes in metabolism, Δq_m , using a proportionality constant $\mu_{bl} = 0.932$ (K^{-1}), which is derived from research by Stolwijk:

$$\Delta\beta = \mu_{bl} \cdot \Delta q_m \quad (3.12)$$

Using $\Delta\beta$, the blood perfusion term in Equation 1 can be rewritten as:

$$\beta(T_{bl,a} - T) \quad (3.13)$$

where

$$\beta = \beta_0 + \Delta\beta \quad (3.14)$$

3.4.5. Numerical methods

In this multi-node regulation model, the computational unit is a facet, with multiple computing nodes assigned to each radial facet. The temperature gradients are typically steepest

near the exterior, making these areas particularly prone to calculation errors. To ensure accurate temperature calculations, a higher number of computing nodes are allocated to the outer layer. The current nodal configuration reflects the values presented from Fiala's research [10].

A finite-difference scheme is used to discretize Equation (3.1), converting the continuous partial differential equation into a solvable discrete form, enabling numerical solution and analysis. Unlike the first-order accuracy of forward or backward difference methods, the central difference method achieves second-order accuracy, leading to more precise results. This model employs an explicit calculation approach, where the node's temperature for the next time step is determined based on the conditions at the current time step.

$$\rho_r c_r \frac{T_r^{(t+1)} - T_r^{(t)}}{\Delta t} = k_r \left(\frac{T_{r+1}^{(t)} + T_{r-1}^{(t)} - 2T_r^{(t)}}{\Delta r^2} + \frac{T_{r+1}^{(t)} - T_{r-1}^{(t)}}{2r\Delta r} \right) + q_{m,r}^{(t)} + \beta_r^{(t)} (T_{bl,a}^{(t)} - T_r^{(t)}) \quad (3.15)$$

The accuracy of the finite-difference method largely depends on how boundary conditions are set. Typically, temperature nodes are placed at interfaces, assuming flat geometry for heat flux calculations. To prevent errors in transient conditions, the last node of a homogeneous tissue layer is connected to an additional virtual node that possesses the same thermophysical properties. Similarly, a virtual node is placed before the first node of the next tissue layer. The boundary condition is satisfied by ensuring that the temperatures and heat fluxes match at the interface between the regular and virtual nodes of adjacent layers. In cylindrical coordinates, the interface temperature T_{ifc} depends on the interface radius r_{ifc} and the temperatures of adjacent nodes T_- (corresponding to the radius $r_- = r_{ifc} - \Delta \frac{r}{2}$) and T_+ (corresponding to the radius $r_+ = r_{ifc} + \Delta \frac{r}{2}$).

$$T_{ifc} = \frac{\ln\left(\frac{r_{ifc}}{r_+}\right)}{\ln\left(\frac{r_-}{r_+}\right)} T_- + \frac{\ln\left(\frac{r_{ifc}}{r_-}\right)}{\ln\left(\frac{r_-}{r_+}\right)} T_+ \quad (3.16)$$

$$q_{ifc} = \frac{k_r(T_- - T_+)}{r_{ifc} \ln\left(\frac{r_+}{r_-}\right)} \quad (3.17)$$

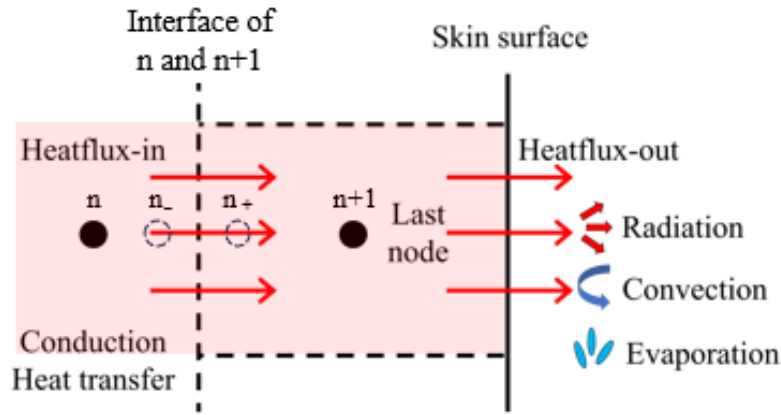


Figure 3.5. Virtual nodes for interface calculation and thermal calculations near the skin

We derive the following equation from the conservation of energy flowing through the interface:

$$\Delta\phi \frac{k_n(T_- - T_+)}{\ln\left(\frac{r_+}{r_-}\right)} = \Delta\phi \frac{k_{n+1}(T_n - T_{n+1})}{\ln\left(\frac{r_{n+1}}{r_n}\right)} \quad (3.18)$$

3.4.6 Core and surface temperature

The model exclusively considers radial heat transfer, which lead to an asymmetrical temperature gradient from the body surface to the core. The axial (central) temperature of each segment is pivotal, serving as a crucial boundary condition for determining the temperatures of each node. Thus, accurately setting the axial temperature boundary condition is paramount importance. In heat transfer calculations, the radius of the previous node serves as the

denominator for determining the heat transfer term of subsequent node. This approach requires consideration of the core axis radius, because neglecting it could result in inaccuracies in the temperature calculations. In this study, the core axis radius is set at 1/10 of the core tissue radius. This specific setting is designed to ensure that the core axis is accounted for, without overwhelming the calculations.

Additionally, the temperature of each node within the tissue layer is influenced by the temperatures of the preceding and subsequent nodes. However, at the last node of each body section, it is impractical to create a virtual next node in the air for continue calculations. Therefore, the temperature is calculated based on the principle of energy conservation applied to the control volume encompassing the last node. Since the outer skin, where the last node is located, has neither blood flow nor metabolic activity, only the thermal conduction term needs to be considered. For each control volume, heatflux-in and heatflux-out at both ends are initially calculated, followed by the determination of the net heat increase using interpolation.

As illustrated in Figure 3.5, heatflux-in represents the heat flow at the interface between the n node and the $n+1$ node, while heatflux-out is derived using the `BOUNDARY_HEAT_FLUX` function in User Defined Function (UDF). These values are then utilized to compute the net heat increase, primarily through the thermal conduction term. Throughout this process, the temperature gradient from the skin to the last node is assumed to change linearly, and Fourier's law of heat conduction is applied to finally calculate the skin temperature. This calculated skin temperature is subsequently used as a boundary condition for heat exchange calculations with the environment.

3.5. Implementation of thermoregulation model through CSP-CFD coupled analysis

The coupled simulation of the thermoregulation model and CFD is a critical component of this research. Using the commercial CFD software ANSYS FLUENT 2022 R2, we integrated the CSP with Fiala model to conduct numerical calculations. We established boundary conditions for all analysis grids on the CSP surface and carried out coupled analyses that include transient calculations of flow fields, temperature fields, and humidity fields. To implement the thermoregulation model, UDF is utilized, enabling users to customize boundary conditions and physical property values. Figure 3.6 illustrates the application and calculation process of the Fiala model using UDF. The `DEFINE_INIT` macro, which defines the initial conditions for the analysis, is particularly important. This macro inputs the initial parameters necessary for Fiala model calculations into each cell at the beginning of the analysis. Therefore, the `DEFINE_INIT` macro operates once at the beginning of the simulation. Conversely, the `DEFINE_ADJUST` macro is applied repeatedly with each iteration of the analysis. For instance, it reads skin surface temperatures from the previous step and averages the skin temperatures of each segment. This approach mirrors the process used in actual thermal manikins, where temperatures from multiple sensors within a single segment are averaged to determine the temperature for that segment. The `DEFINE_PROFILE` macro sets the boundary conditions for analysis, including skin surface temperature and humidity distribution as calculated from the Fiala model. Additionally, the `DEFINE_DIFFUSIVITY` macro defines the diffusion term for the user-defined scalar transport equation utilized in the analysis. In this study, humidity is treated as a passive scalar, and the analysis of the humidity field is conducted using the specified scalar transport equation. The diffusion coefficient within the diffusion term employs a formula for the diffusion coefficient of water vapor in air, calculated based on the temperatures of each cell [13].

The sequence of calculations illustrated in Figure 3.6 is engineered to perform iterative calculations (Looping) across all faces (surface meshes) and cells (volume meshes) within the analysis area. Once the iterative calculations for all meshes are completed, the process advances to the next iteration to conduct new calculations.

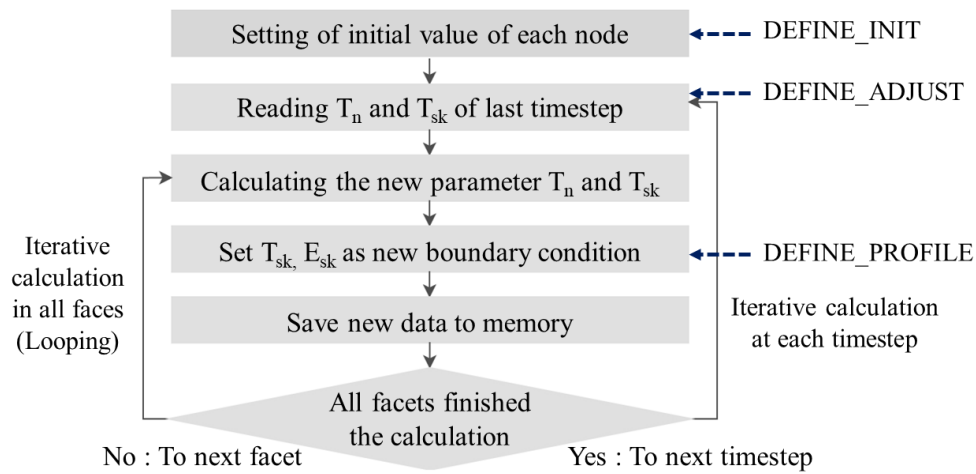


Figure 3.6. Calculation process of the Fiala thermoregulation model using UDF

REFERENCES

- [1] H. Xing, A. Hatton, H.B. Awbi, A study of the air quality in the breathing zone in a room with displacement ventilation, *Build. Environ.* 36 (2001) 809–820.
- [2] H. Brohus, K.D. Balling, D. Jeppesen, Influence of movements on contaminant transport in an operating room., *Indoor Air.* 16 (2006) 356–372.
- [3] S. Murakami, S. Kato, J. Zeng, Flow and temperature fields around human body with various room air distribution, Part 1-CFD study on computational thermal manikin, *ASHRAE Trans.* 103 (1997) 3–15.
- [4] S. Murakami, S. Kato, J. Zeng, Combined simulation of airflow, radiation and moisture transport for heat release from a human body, *Build. Environ.* 35 (2000) 489–500.
- [5] C. Topp, P.V. Nielsen, D. Sorensen, Application of computer simulated persons in indoor environmental modeling/Discussion, *ASHRAE Trans.* 108 (2002) 1084–1084.
- [6] D.N. Sørensen, L.K. Voigt, Modelling flow and heat transfer around a seated human body by computational fluid dynamics, *Build. Environ.* 38 (2003) 753–762.
- [7] N.P. Gao, H. Zhang, J.L. Niu, Investigating indoor air quality and thermal comfort using a numerical thermal manikin, *Indoor Built Environ.* 16 (2007) 7–17.
- [8] P.V. Nielsen, S. Murakami, S. Kato, C. Topp, J.-H. Yang, Benchmark tests for a computer simulated person, Aalborg University, *Indoor Environmental Engineering.* (2003).
- [9] Y. Tao, W. Yang, K. Inthavong, J. Tu, Indoor particle inhalability of a stationary and moving manikin, *Build. Environ.* 169 (2020) 106545–106545.
- [10] D. Fiala, K.J. Lomas, M. Stohrer, A computer model of human thermoregulation for a wide range of environmental conditions: the passive system, *J. Appl. Physiol.* 87 (1999) 1957–1972.
- [11] H.H. Pennes, Analysis of tissue and arterial blood temperatures in the resting human forearm, *J. Appl. Physiol.* 1 (1948) 93–122.
- [12] C. Wyistdham, J. Morrison, C. Williams, N. Strydom, M. Von Rahden, L. Holdsworth, C. Van Graan, A. Van Rensburg, A. Joffe, A. Heyns, Inter-and intra-individual differences in energy expenditure and mechanical efficiency, *Ergonomics* 9 (1966) 17–29.
- [13] D.M. Gates, *Biophysical ecology*, Springer Science & Business Media, 2012.

Validation of CSP's thermal prediction accuracy

4.1. Introduction

This chapter deals with validation procedure for the thermal prediction accuracy of the newly developed CSP. To validate the thermal prediction accuracy of developed CSP integrated with multi-node thermoregulation model, we performed CSP-CFD coupled analysis corresponding chamber experiment using ANDI thermal manikin. The skin surface temperatures measured by experiment and analyzed by CFD simulation were compared.

4.2. Chamber experiment using ANDI thermal manikin

As shown in Figure 4.1, the ANDI thermal manikin was positioned at the center of a chamber room with the dimensions $1.4 \times 2.1 \times 2.4$ m. In front of the manikin, a 0.7 m wide desk was positioned at 0.7 m off the floor, adjacent to the front wall. The thermal manikin was set in three different states according to clothing status and posture: naked/standing, clothed/standing, and naked/seated.



Figure 4.1. Analysis domain corresponding to the chamber experiment

To maintain the indoor temperature within the chamber, the air conditioning system was controlled. The airflow rate of both supply air (SA) and return air (RA) are $164 \text{ m}^3/\text{h}$, while that of the outdoor air (OA) is $47 \text{ m}^3/\text{h}$. The chamber's ambient temperature is regulated using both OA and SA, with the RA temperature as the indoor temperature control point. To maintain indoor pressure, EA is released through the gap between the door and the floor of the chamber. In the experiment, the temperature and relative humidity of the SA, OA, and RA were sampled for 30 min; the skin surface temperature of the thermal manikin was also sampled. At the same time, the temperature change in the chamber wall surface over time was recorded. After 30 minutes, the thermal imaging camera recorded the body surface temperature distribution of the thermal manikin.

4.3. CSP-CFD coupled analysis corresponding to chamber experiment

The analysis domain and boundary conditions for the CFD simulation were defined to replicate the previously introduced chamber experiment. Figure 4.2 illustrates the grid design around the CSP using the clothing model. Significant differences were observed in the analysis mesh around the CSP, depending on the clothing worn. The domain with the naked CSP was discretized with 2.6 million polyhedral cells, while the domain with the clothed CSP with 7.2 million polyhedral cells. To ensure the quality of CFD simulation involving a CSP, recommended analytical methods from a previous study, including grid design in the analytical domain, were adopted [1]. The thickness of boundary meshes, cell skewness, and growth ratio were carefully controlled.

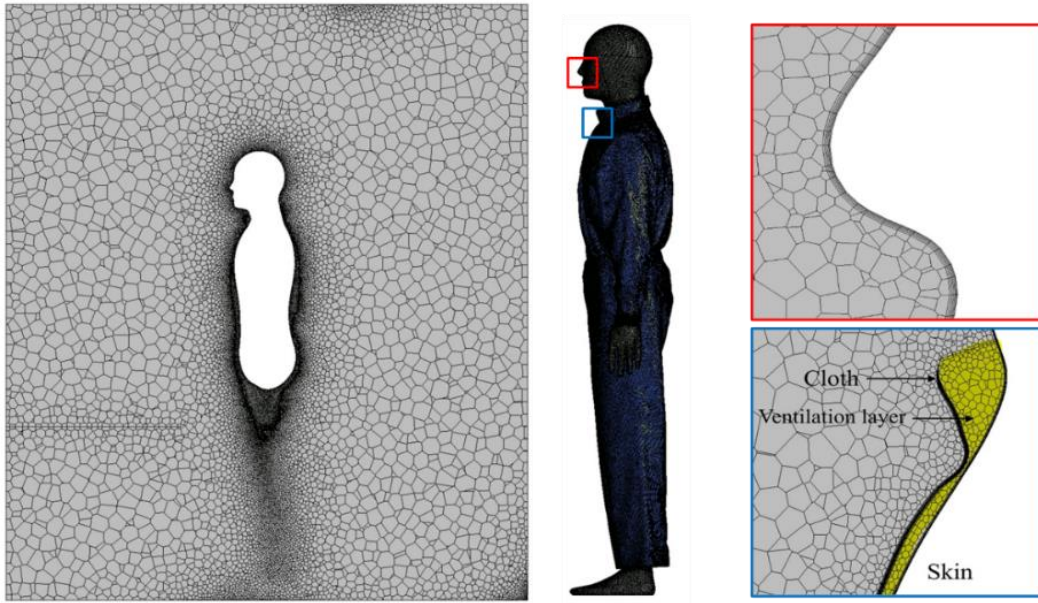


Figure 4.2. Grid design around CSP with the clothing model

To accurately represent the narrow ventilation layer between the clothing and skin, as well as the complex shape of the clothing, we applied five layers of boundary cells with a thickness of approximately 0.2 mm to the surfaces of the clothing and CSP. This configuration enables the precise prediction of airflow and heat transfer within the viscous sublayer of the CSP. Consequently, the dimensionless wall distance (y^+) remained below 1.0 over the entire surface of the cloth and CSP. With these efforts, we very carefully conducted grid independence check, and the number of grid and the final grid design around CSP were determined based on the results of this check.

Table 4.1 details the analytical and boundary conditions for the CFD. Clothing properties were configured based on polyamide cloth. The CFL (Courant-Friedrichs-Lewy) number was set to show values below 1 throughout the domain, except for areas near the inlet with high velocities, by setting the time step interval to less than 0.2 s. The convergence criteria were set with a scaled residual of continuity less than 10^{-4} and for the other equations less than 10^{-5} . The buoyancy

effect was calculated based on the Boussinesq approximation. In addition to convective heat transfer through CFD analysis, the radiative heat transfer was analyzed using Surface-to-Surface(S2S) model with a view factor calculation [2]. In the experiment, although the temperature of RA was intended to be maintained at the indoor set point temperature, slight variations was observed in the measured results of the RA temperature and relative humidity over time. Hence, the profiles of temperature and humidity from SA, OA, and RA over time were extracted and applied to the boundary conditions for accurate reproduction of experimental conditions.

Table 4.1. Analytical and boundary conditions

Algorithm	SIMPLE (Unsteady calculation)
Turbulence model	SST k- ω model
Discretization scheme	Second-order upwind scheme for convection
Inflow boundary	OA: Velocity inlet, 47 m ³ /h, Turbulent intensity = 10% SA, RA: Velocity inlet, 164 m ³ /h, Turbulent intensity = 10% Measured temperature/humidity-time profile
Outflow boundary	Gradient zero
Wall treatment (Room)	Velocity: No slip, Temperature: measured temperature-time profile
Wall treatment (CSP)	Velocity: No slip Temperature: Calculated by Fiala thermoregulation model Humidity: water vapor generation assuming 6 % of skin wettedness
Radiation model	Surface to Surface Model (S2S model) with view factor calculation Emissivity: skin = 0.98, cloth = 0.95
Properties of cloth	Specific heat capacity: 1850 J/kg·K Density: 450 kg/m ³ Thermal conductivity (λ): 0.06 W/m·K

Table 4.2. Experimental and analytical cases.

	Indoor setpoint temperature	Clothing condition	Posture of CSP
Case-1A	$T_{RA} = 28\text{ }^{\circ}\text{C}$	Naked	Standing
Case-1B		Clothed	
Case-1C		Naked	Seated
Case-2A	$T_{RA} = 24\text{ }^{\circ}\text{C}$	Naked	Standing
Case-2B		Clothed	
Case-2C		Naked	Seated
Case-3A	$T_{RA} = 18\text{ }^{\circ}\text{C}$	Naked	Standing
Case-3B		Clothed	
Case-3C		Naked	Seated

In the experiments and CFD analysis, nine analytical cases were prepared according to the setpoint temperature, the posture of the manikin, and the presence or absence of clothing, as summarized in Table 4.2. In addition to the multifaceted examination of the reproducibility of the thermal manikin experiment, the impacts of environmental conditions, clothing, and human posture on the thermoregulation and microclimate surrounding the CSP were discussed through parametric numerical analyses under varying conditions.

4.4. CFD analysis results and discussion

4.4.1. Airflow and temperature distribution

Figure 4.3 and 4.4 presents the distributions of the scalar velocity and temperature in the indoor space for all the analytical cases. The analysis revealed non-uniform airflow pattern with variation for each case owing to differences in clothing conditions, CSP posture, and temperature/humidity of the supplied air. Overall, noticeable downward airflow from the OA to the ceiling was observed.

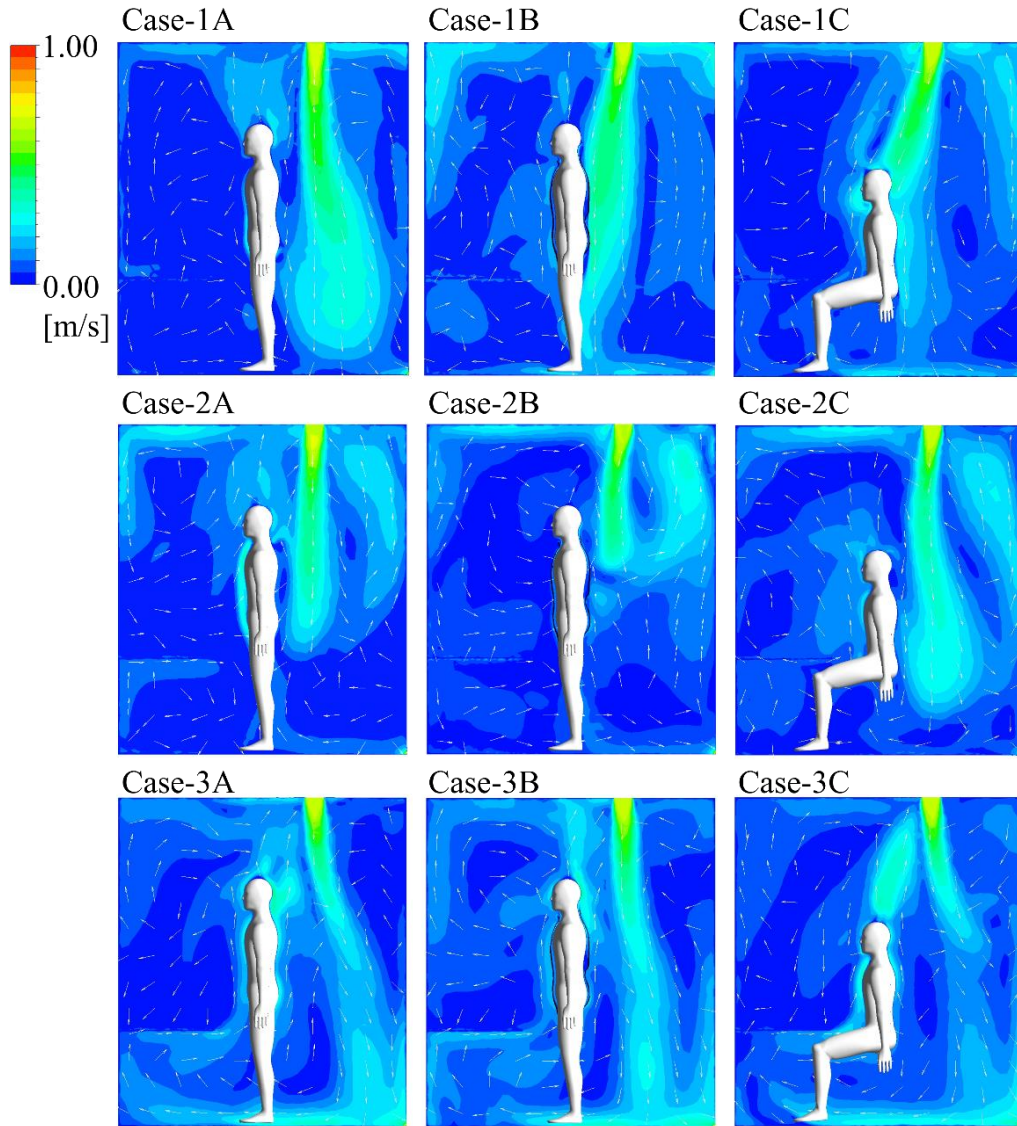


Figure 4.3. Indoor scalar velocity distribution analysis results at $t = 1,800$ s for 9 cases depending on indoor setpoint temperature, clothing condition, and posture. A: Naked standing; B: Clothed standing; C: Naked seated.

Upon reaching the floor or CSP, an upward airflow was observed directed towards the RA in the ceiling, whereas an airflow was observed heading towards the EA at the lower part of the back wall. Case B, with clothing in a similar environment, exhibited a more complex airflow distribution than Case A. This is believed to be because the complex geometry of the clothing affected the airflow around the CSP. However, given the proximity of the two inlets, the airflow from the OA is significantly influenced by the strong airflow from the SA, making it difficult to definitively characterize the airflow properties for each case.

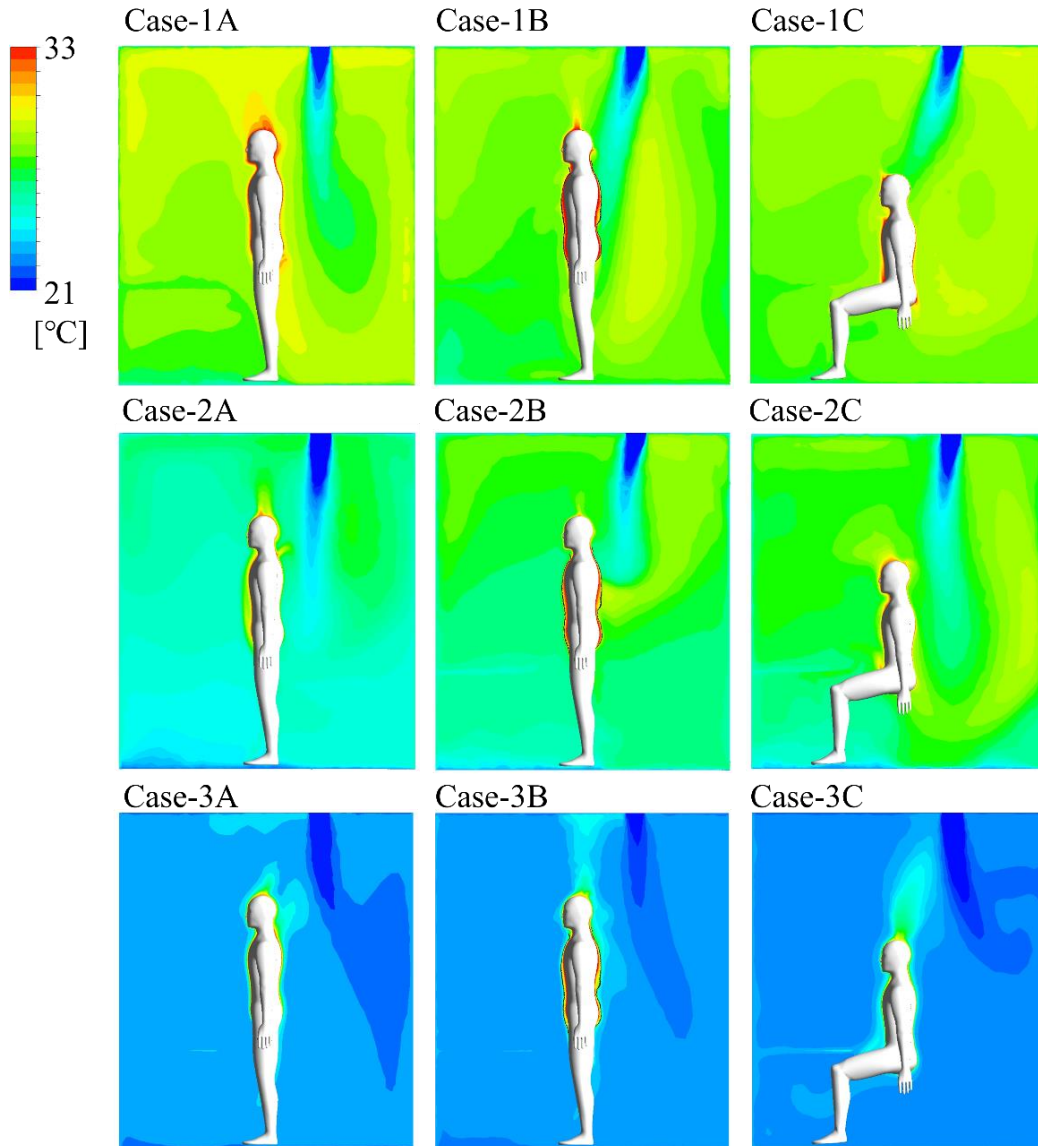


Figure 4.4. Temperature distribution analysis results at $t = 1,800$ s for 9 cases depending on indoor setpoint temperature, clothing condition, and posture. A: Naked standing; B: Clothed standing; C: Naked seated.

Cases 1B and 1C show that the current of the OA directly reached the CSP owing to the influence of the SA. Furthermore, considering the posture of the CSP, the impacts of the SA and OA were relatively greater in Case C. The temperature distribution exhibits a tendency like that of the flow field. In most cases, a thermal plume is observed at the top of the CSP head. However, it was not clear whether the RA was located on the ceiling; therefore, it was an updraft due to air circulation or buoyancy effect.

4.4.2. Clothing impact

To investigate the impact of clothing on the microclimate around the CSP, the distributions of airflow, temperature, and absolute humidity around the CSP in Cases 1A and 1B were analyzed, and the results are summarized in Figure. 4.5. In the case of clothed CSP, the diffusion of temperature and humidity on the CSP surface was inhibited owing to the temperature resistance between the cloth and skin, leading to an elevated distribution of temperature and humidity. The airflow also generally showed stagnant airflow, except for the relatively high velocity airflow around the neck and shoulders owing to the narrow opening in the ventilation layer between the clothing and skin.

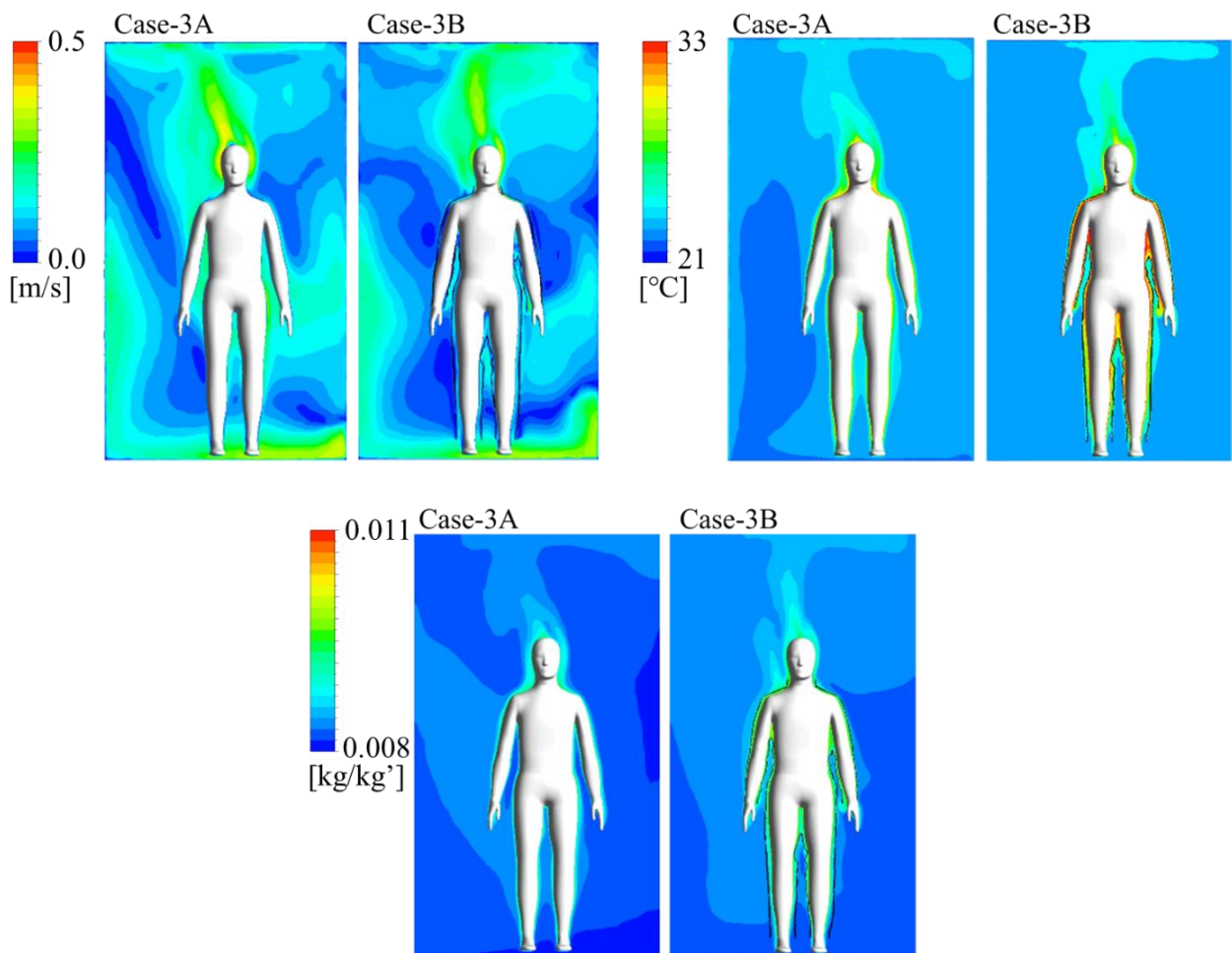


Figure 4.5. Velocity, temperature, humidity distribution around CSP at $t = 1,800$ s in Case-3A and Case-3B

This led to an increase in the skin surface temperature, as shown in Figure. 4.6. When comparing Case A and B, a clear temperature increase was observed in the clothed skin compared with the exposed skin temperature. Even when the surrounding environment temperature was lowered to a similar level, the body temperature decreased by about 1.6 °C in case B, whereas it decreased by about 2.0 °C in case A. Additionally, the relatively lower skin temperature in peripheral body parts, such as the feet and hands, was well reproduced by the characteristics of the body temperature distribution shown in the thermal manikin.

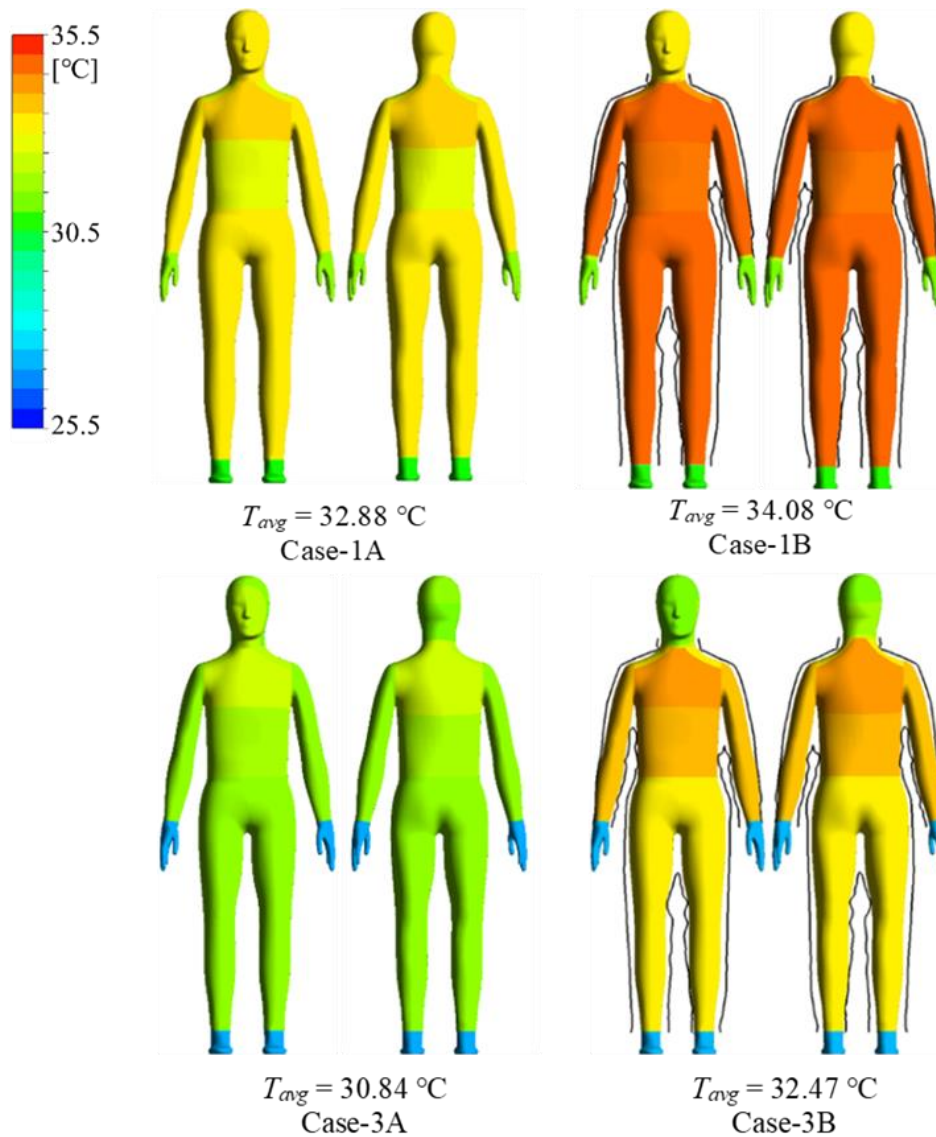


Figure 4.6. Skin temperature distribution of CSP at $t = 1,800$ s in four cases. A: Naked standing; B: Clothed standing

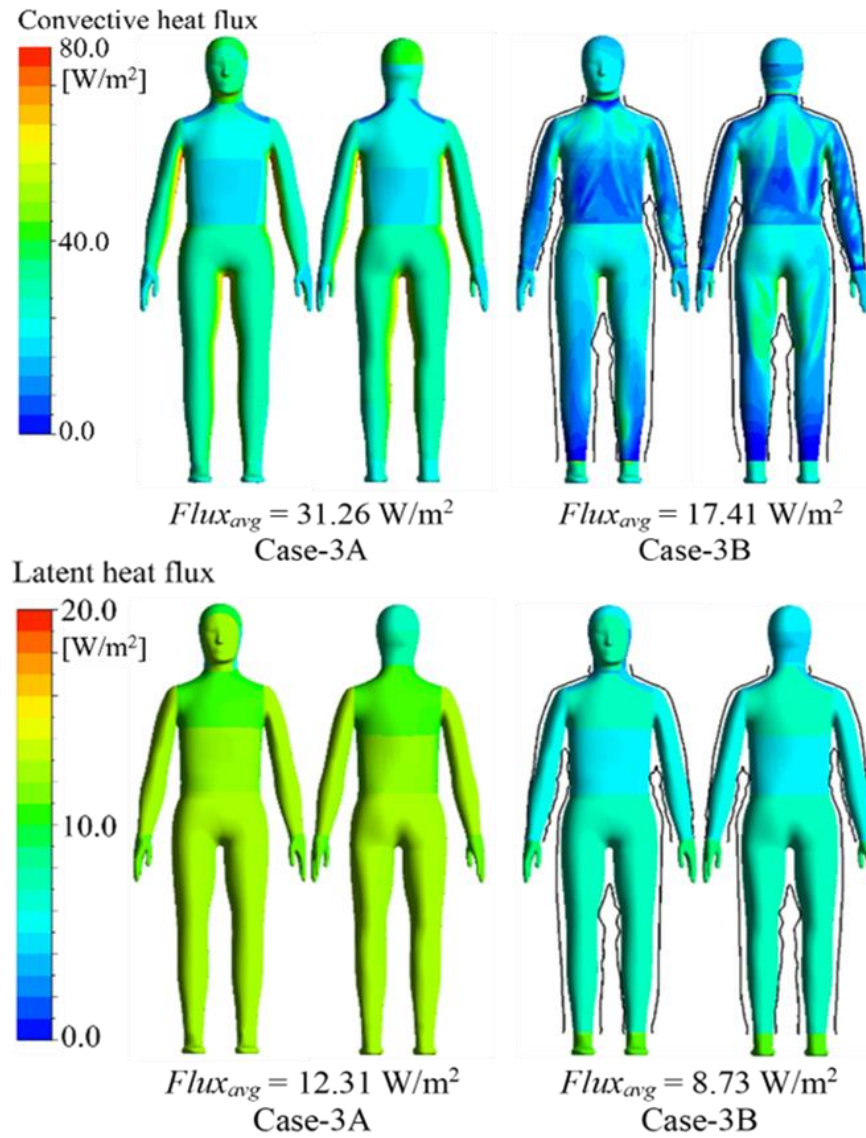


Figure 4.7. Convective and latent heat flux distribution on the skin surface of CSP at $t = 1,800 \text{ s}$ in Case-3A and Case-3B. A: Naked standing; B: Clothed standing

Furthermore, the convective and latent heat fluxes owing to the stagnant flow pattern and increased humidity distribution in the air gap between the clothing and skin can also be observed in Figure. 4.7. Simultaneously, because convective heat loss is highly influenced by the distribution of adjacent airflows, the average adjacent airflow velocities were measured in the clothed parts in Case 3B and in the same parts in Case 3A. In Case 3B, the airflow velocity decreased in most clothed parts (thorax: 0.086 m/s, abdomen: 0.047 m/s, arm: 0.024 m/s, leg: 0.038 m/s) compared with Case 3A(thorax: 0.134 m/s, abdomen: 0.078 m/s, arm: 0.063 m/s, leg:

0.081 m/s), except for the shoulders. The shoulder was narrow and close to the opening of the ventilation layer between the cloth and the skin, so the airflow velocity increased from 0.03 m/s (Case 3A) to 0.133 m/s in (Case 3B). Consequently, the convective heat flux decreased in most clothed parts but increased in the shoulder. Additionally, the latent heat flux decreased in the parts covered by clothing owing to the increase in humidity between the clothing and the skin.

The role of clothing in thermal resistance is demonstrated in Figure 4.8 and 4.9, where Cases 3A and 3B compare the effects of clothing under identical environmental conditions. The total thermal resistance for each body segment was calculated using Equations (4.1) and (4.2). In these equations, R ($\text{m}^2 \cdot \text{K}/\text{W}$) represents thermal resistance, α ($\text{W}/\text{m}^2 \cdot \text{K}$) the combined coefficient of heat transfer from convection and radiation, λ ($\text{W}/\text{m} \cdot \text{K}$) the thermal conductivity of the clothing material, and n (m) the thickness of the cloth. Although actual clothing thickness was not modeled, a uniform thickness of 0.003 m was assumed for the calculations.

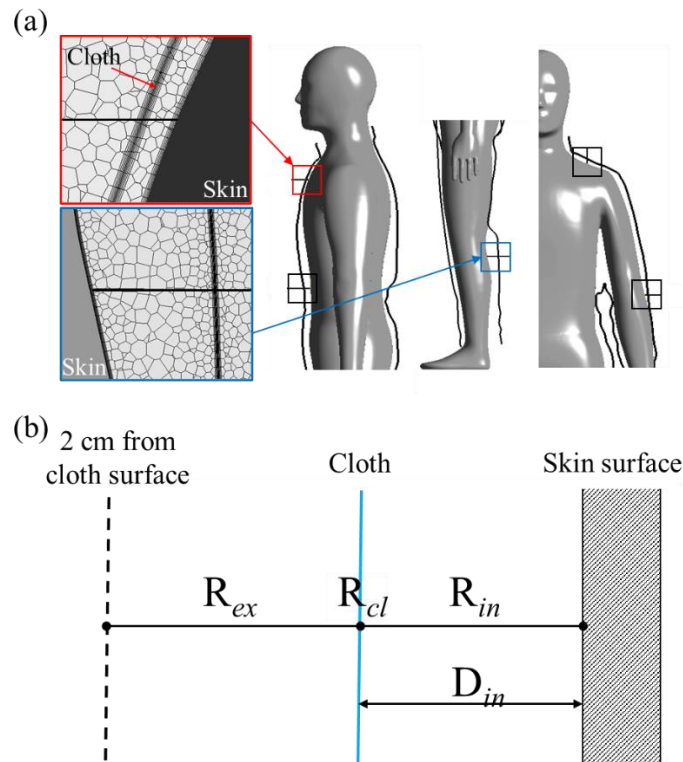


Figure 4.8. (a) Location of analysis target regions in each segment. (b) Outline of the thermal resistance analysis.

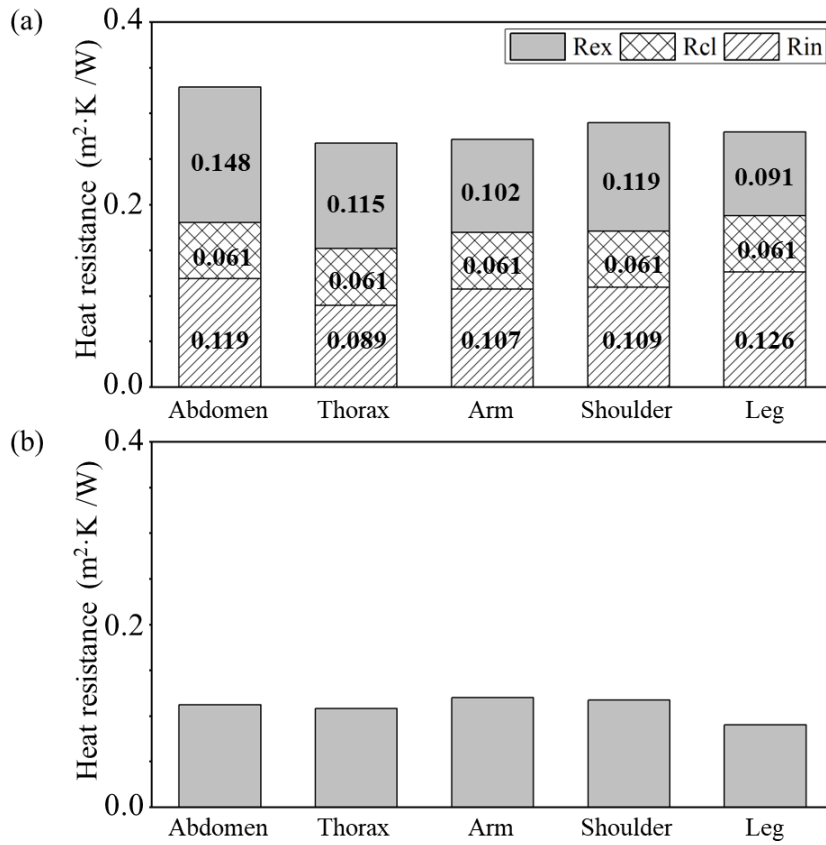


Figure 4.9. (a) Calculation results of thermal resistance for clothed condition (Case 3B), and (b) naked condition (Case 3A).

$$R = \frac{1}{\alpha} \text{ or } \frac{n}{\lambda} \quad (4.1)$$

$$R_{total} = R_{ex} + R_{cl} + R_{in} \quad (4.2)$$

$$y^+ = \frac{u^* y_1}{\nu_a} \quad (4.3)$$

$$u^* = \sqrt{\frac{\tau_w}{\rho}} \quad (4.4)$$

The total thermal resistance, R_{total} , as defined in Equation (4.2), was calculated by considering the thermal resistance of the ventilation layer, the interior of the clothing, and the exterior side of the clothing. Figures 4.9 (a) and (b) show that thermal resistance was higher in areas covered by clothing compared to uncovered areas, reflecting the role of clothing as a barrier that shows heat transfer. In the case with clothed CSP, the total thermal resistance in each segment saw substantial

increases: 331.2% in the abdomen, 313.7% in the thorax, 276.6% in the arm, 226.5% in the shoulder, and 307.1% in the leg, relative to the naked condition. When converted to clo units (1 clo = 0.155 m²·K/W), these increases correspond to 1.39 clo for the abdomen, 1.01 clo for the thorax, 1.14 clo for the arm, 0.92 clo for the shoulder, and 1.13 clo for the leg in Case 3B. Additionally, R_{ex} exhibited significant variations depending on the temperature and airflow around the clothing. Conversely, areas with greater distances between the clothing and the skin displayed higher R_{in} (m²·K/W) values. For the arm and shoulder, which maintained similar distances from the skin to the clothing surface, the resistance was comparable. This highlights a strong correlation between the thickness of the ventilation layer and thermal performance, emphasizing the importance of precise clothing geometry in thermal studies.

4.4.3. Comparison of skin temperature between CSP and thermal manikin

The reproducibility of the thermoregulatory analyses using CSP was examined by comparing the simulated skin surface temperature with that measured in the chamber experiments. Figure. 4.10 shows the skin and cloth surface temperatures of the manikin and the CSP at $t = 1,800$ s for Cases 1A and 1B. The surface temperature of the naked/clothed thermal manikin was measured at distance of 2 m using a thermal imaging camera (Testo 882, measurement accuracy of ± 2 °C).

In Case 1A, with the naked CSP, similarities in the temperature level and distribution were observed, and the characteristics of the hands and feet showing relatively lower temperatures than the other parts were well reproduced. In Case 1B, with the clothed CSP, a relatively high temperature in the skin-clothing contact region at the thorax and armpit was also observed. However, sufficient verification was not possible with the visual analysis method using the temperature distribution contours.

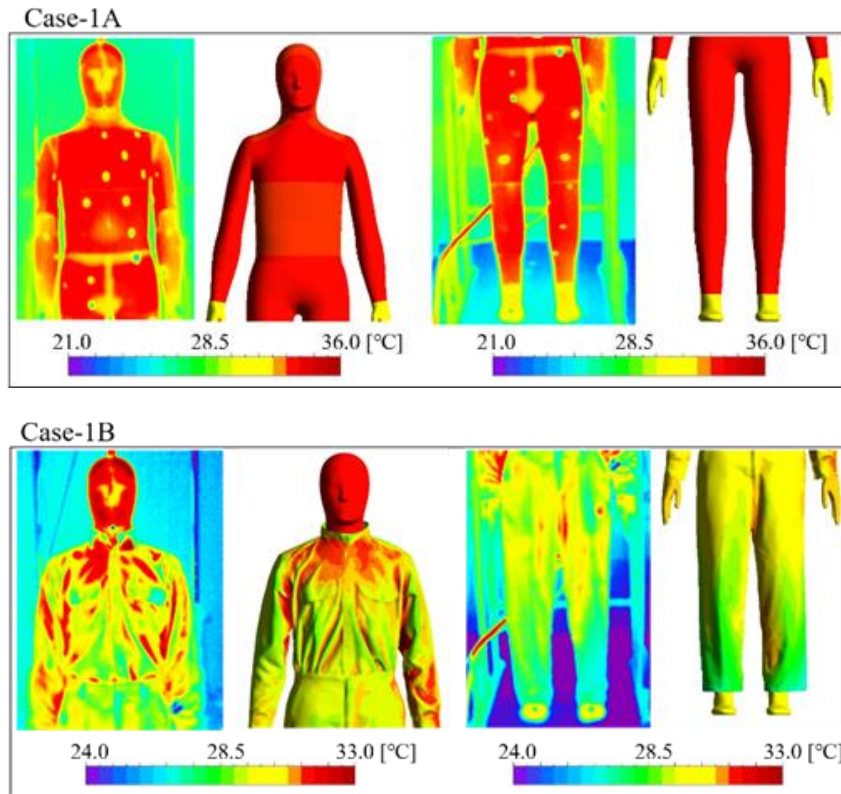


Figure 4.10. Skin temperature and cloth surface temperature of manikin and CSP at $t = 1,800$ s in Cases 1A and 1B. A: Naked standing; B: Clothed standing.

Hence, the average skin surface temperatures of the thermal manikin and the CSP after 30 min of simulation were compared (Figure. 4.11). Just as the skin temperature of the manikin was well maintained in the given temperature and humidity environments, the skin temperature of the CSP was well represented. In Cases A and B, the phenomenon of the manikin's average skin temperature rising by more than 1.0 °C in similar environmental conditions due to wearing clothes was well reproduced in CSP. In Cases 1A and 1B, the temperature of the manikin gradually increased in response to the progressive rise in ambient temperature, which the CSP replicated well. In Case 2, there were significant fluctuations in ambient temperature, but both the manikin and the CSP adequately managed the temperature rises and falls. In Case 3, the ambient temperature control was well-maintained, allowing for verification of the temperature maintenance capabilities of both the manikin and the CSP.

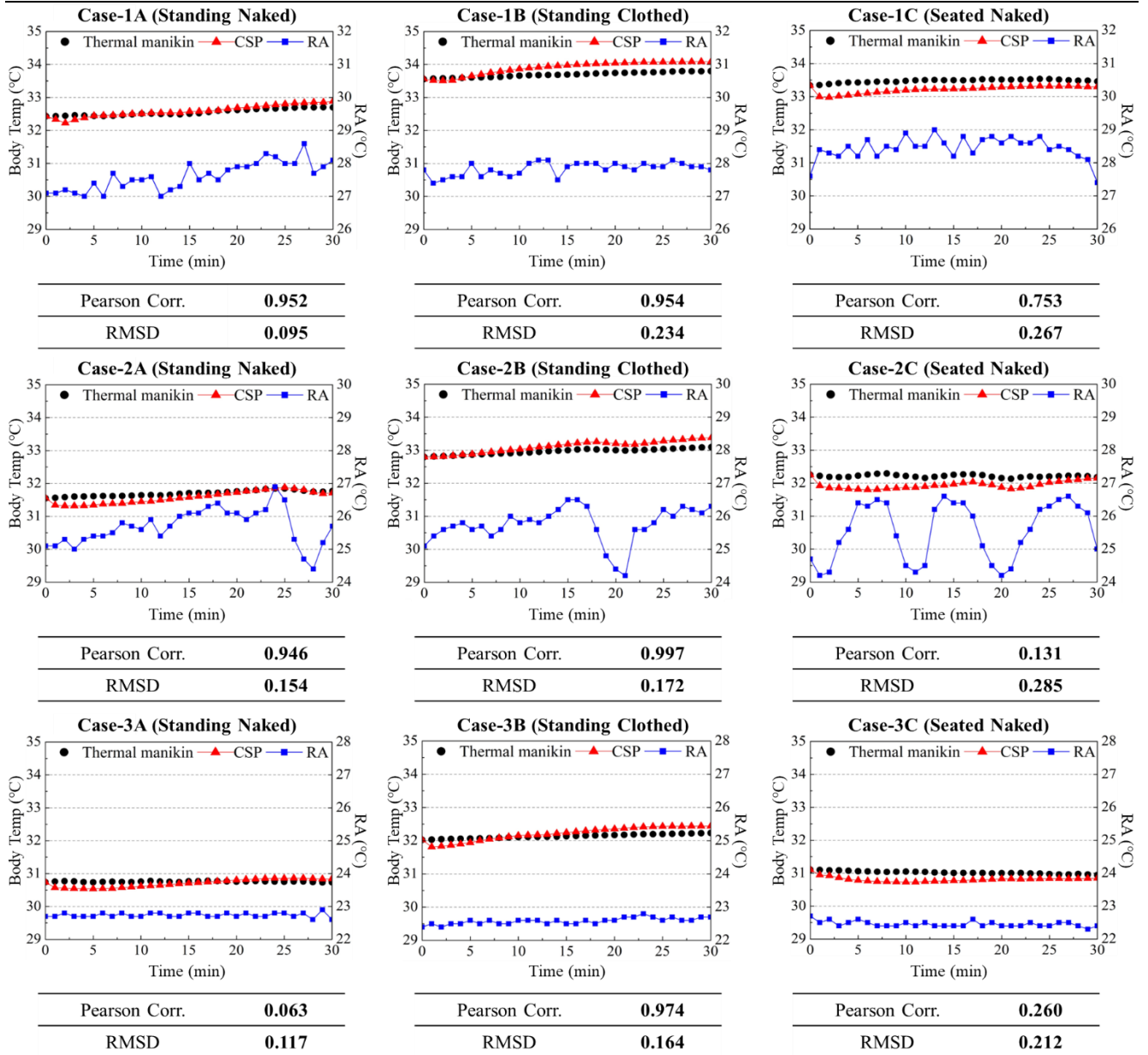


Figure 4.11. Comparison of average body temperature of manikin and CSP in all cases.

The difference between the average skin temperature of the thermal manikin and the CSP was quantified by root mean square deviation (RMSD) and Pearson correlation coefficient. RMSD was used to quantitatively assess the absolute discrepancies between the experimental and simulated temperatures, directly measuring the model's predictive accuracy. Meanwhile, the Pearson correlation coefficient evaluated the strength of the linear relationship between the two datasets, indicating how well the model captures the pattern of the experimental data. These metrics offer a comprehensive understanding of the model's accuracy and consistency.

In all cases, the difference between the average skin temperature of the manikin and the CSP over 30 min was greatest in the order of C, B, and A, under similar chamber environmental conditions. This is due to the difference in clothing representation between Cases A and B, as factors such as clothing permeability and moisture movement were not applied, which could have led to a difference in error. Furthermore, in Case C, it was assumed that the airflow supplied to the chamber directly influenced the head of the CSP, which contributed to the error difference. However, across all cases, the average error and RMSD value between the body temperatures of the thermal manikin and the CSP were below 0.3. This was within the acceptable range for thermal comfort assessment. The Pearson correlation coefficient measures the strength of a linear relationship between two variables and ranges between -1 and +1. Values close to +1 or -1 indicate a perfect positive or negative linear relationship, respectively, while values near zero signify no linear relationship. The threshold for good consistency in research or analysis can vary. In some contexts, a correlation coefficient above 0.5 might be considered strong, but for this study, which aims to precisely replicate the thermal functions of a thermal manikin, a higher threshold of 0.75 was used to assess consistency. In most cases using the standing CSP, the correlation coefficient was above 0.9, showing very strong consistency. However, in Case 3A, due to minimal changes in ambient temperature and repeated minor temperature fluctuations of the manikin, the correlation coefficient was nearly zero. In such cases, it is inappropriate to judge the lack of consistency solely based on the Pearson correlation coefficient. RMSD of 0.117 indicated that the predictive error was sufficiently small, confirming that the standing CSP has sufficient reliability in predicting thermal accuracy.

However, the seated CSP didn't achieve the sufficient correlation coefficients even under conditions of significant ambient temperature changes. The experimental setup involved a

confined chamber with the presence of SA, RA, OA, and EA, which resulted in a complex indoor environment. Additionally, the proximity of the CSP to SA and OA location means that even minor positional differences between the manikin and the CSP can lead to significant errors. Specifically, in cases with the seated CSP, the inflow air was directed at the head of the CSP, which could introduce additional variables affecting the accuracy of replicating the experimental conditions. Therefore, further validation in simpler environments is necessary for the utilizing the seated CSP.

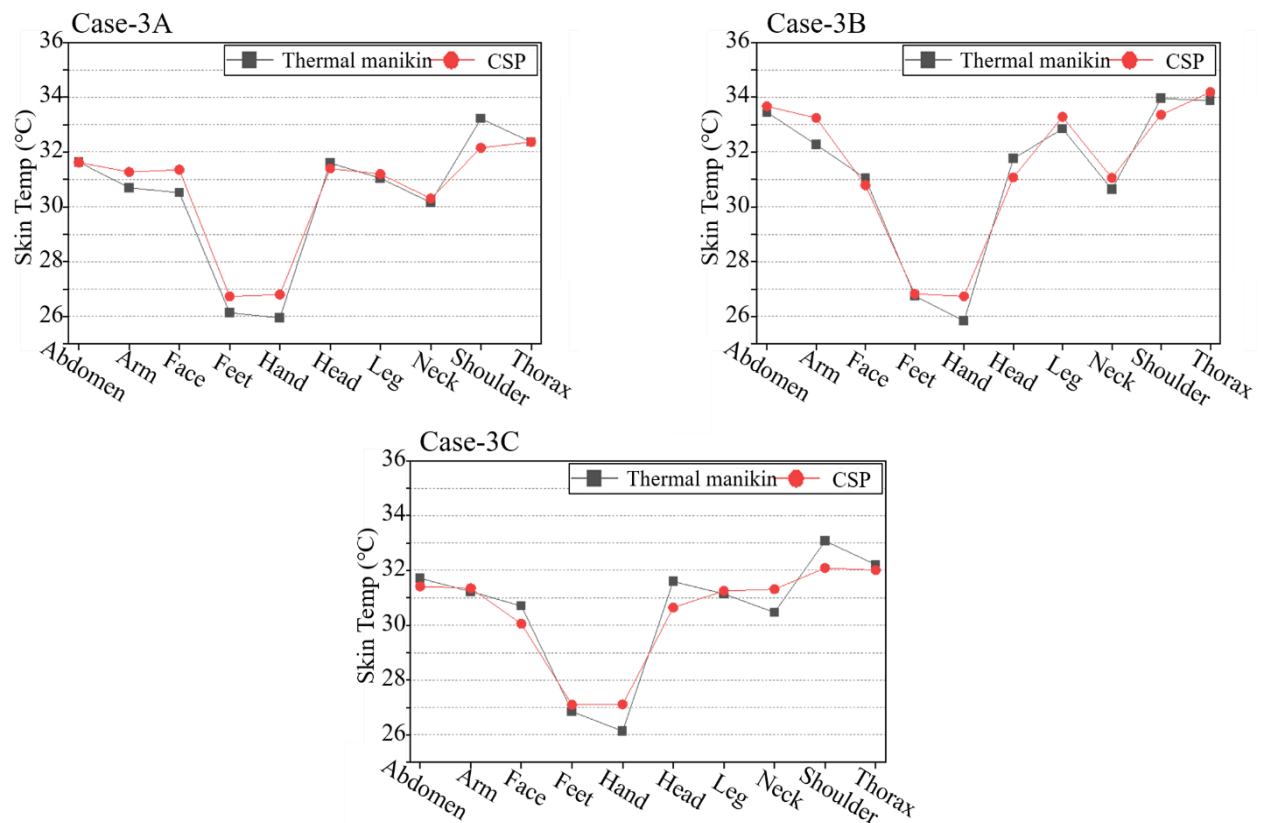


Figure 4.12. Comparison of each segment's skin temperature at $t = 1,800$ s in Cases 3A, 3B, and 3C. A: Naked standing; B: Clothed standing; C: Naked seated.

In Cases 3A, 3B, and 3C, where the difference between the body temperature and ambient temperature was the most significant, the temperatures of each segment of the manikin and the CSP are compared in Figure 4.12. In Case 3A, with the naked standing CSP, a relatively large

temperature difference was observed in parts with small areas or complex geometries, such as the shoulders, face, and hands. However, other parts exhibited similar temperature distributions with temperature differences less than 0.5 °C. In Case 3B with the clothed CSP, the temperature of the covered skin showed a greater increase in the CSP than that in the thermal manikin. Consequently, not only the hands and shoulders but also the arms showed relatively larger temperature differences compared to other areas. In Case 3C with the naked CSP, relatively large temperature differences were observed at the shoulders, hands, neck, head, and face.

This was presumed to be due to the significant impact of the airflow from the inlet on the facial area, which acted as an additional variable, unlike in other cases. Despite this, the CSP accurately replicated the overall temperature distribution trends of the thermal manikin. However, in the case of the seated CSP, there were differences in the temperature distribution around the head area, indicating that further validation is needed. Through this review, it was determined that the digital twin of the thermal manikin developed in this study possesses sufficient thermal prediction capabilities for indoor thermal evaluation analysis. However, the seated CSP showed some deficiencies compared to the standing CSP, indicating the need for further validation and improvement.

The advanced CSP and clothing models developed and validated in this study facilitated a more precise and comprehensive evaluation of indoor environments and human thermal comfort. Notably, they allow for the assessment of the microclimate of the area between the cloth and skin, which is difficult to analyze via conventional measurements. This can contribute significantly to the design of human-centered indoor environments. In future work, it will be necessary to conduct a comparative analysis of advanced CSP with clothing model developed in this study and the previous thermoregulation model with mathematical clothing model.

4.5. Limitation

This research also has some limitations. The CSP integrated with clothing applied in this study could not replicate the contact between the clothing and skin due to technical constraints in grid generation. In real-life situations, contact occurs at the shoulders, upper chest, and buttocks, which can obstruct airflow or create contact thermal resistance. These factors are believed to have influenced the temperature comparison errors in cases where the clothed CSP was utilized, suggesting a potential overestimation of the ventilation rate in the clothing's air layer. Additionally, the porosity of the clothing material was not considered, leading to an assumption that airflow through the clothing was completely obstructed. Shivering and sweat movement were also not accounted for in this study, as their impacts are minimal under the experimental conditions. However, these factors should be critically considered in extreme cold or hot conditions. The study also only developed clothing models for long sleeves and pants. While material-based characteristics can be modified, there is a clear need for more diverse clothing models, such as short sleeves and shorts, to cover a broader range of scenarios. Further research on these topics is considered necessary to develop a more advanced CSP.

4.6. Conclusion

In Chapter 3 and 4, we developed an advanced CSP for CFD analysis targeting a real thermal manikin control using the Fiala multi-node thermoregulation model. To confirm the reproducibility of the thermal manikin, *in silico* experiments corresponding to chamber experiment using a real manikin were conducted, and the prediction accuracy of the CSP was verified by comparing the analysis results with those of the thermal manikin and CSP under the same environmental conditions. Additionally, a clothing model incorporating a detailed

ventilation layer structure (air gap between the skin and cloth) was considered. The average RMSD for the temperature difference between the thermal manikin and the CSP was below 0.285 °C for all cases, which falls within the acceptable range for thermal comfort assessment. The Pearson correlation coefficient for the standing CSP was above 0.9, demonstrating sufficient consistency. However, the seated CSP showed somewhat insufficient consistency in the Pearson correlation coefficients. Furthermore, it was confirmed that the digital twin of the thermal manikin accurately reflects the actual functions of the thermal manikin. The influence of the ventilation layer between the clothing and skin on thermal resistance was confirmed, emphasizing the necessity of accurately modeling clothing. In this study, we developed an advanced CSP that allows parametric analysis under various environmental conditions and scenarios. Using this CSP, the data acquired from precise indoor environmental assessments can provide foundational information and tools for designing human-centered indoor environments.

REFERENCES

- [1] S. Yoo, K. Ito, Validation, verification, and quality control of computational fluid dynamics analysis for indoor environments using a computer-simulated person with respiratory tract, *Jpn. Archit. Rev.* 5 (2022) 714–727.
- [2] ANSYS Fluent 2021 R1 user's guide and theory manual, Ansys Fluent Inc. (2021).

CHAPTER 5

Human breathing

5.1. Introduction

In this chapter, we integrated a numerical nasal cavity model into the developed CSP to replicate the crucial human function of breathing. This integration includes a transient breathing cycle model that simulates variable flow rates over time, enabling dynamic numerical analysis. Notably, in the context of breathing representation, following global pandemics such as SARS-CoV-2, the identification of breathing zones (BZs) has emerged as a significant public health issue. It is crucial for tracking airborne transmission via respiratory aerosol and for establishing guidelines on proper social distancing [1, 2]. Although a BZ lacks a universally accepted definition, it conventionally refers to the area around an individual's nose and mouth—the space where airborne particles and contaminants are most likely inhaled.. Consequently, this chapter explores the use of Scale for Ventilation Efficiency (SVE) levels 4 and 5 to analyze BZs both quantitatively and qualitatively. This analysis is essential for evaluating contaminant transfer and assessing human exposure.

5.2. Integration of CSP with airway model

To precisely analyze the impact of human breathing on the indoor environmental climate and inhalation exposure to indoor pollutants, integrating a numerical airway model that accurately replicates the geometric shape is necessary. Therefore, in this study, the developed CSP model and a nasal cavity model were integrated, as illustrated in Figure 5.1. The integrated numerical

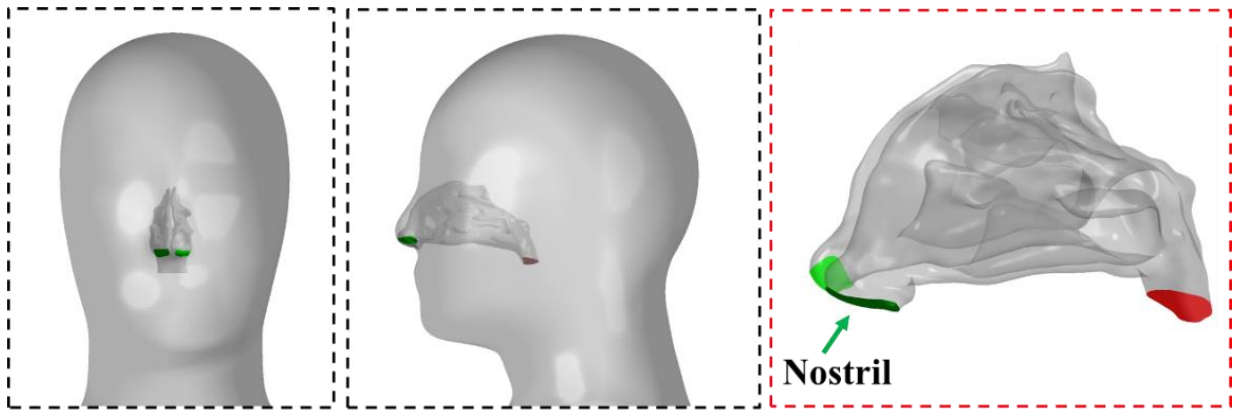


Figure 5.1. CSP integrated with a nasal cavity model

nasal cavity model was developed using CT data obtained from a healthy adult male. Its efficacy in predicting human airway flow has been validated through comparison of airflow distribution with an experimental airway model. This experimental model was created using 3D printing based on the same CT data [3, 4].

5.3. Transient breathing model

A transient breathing profile was applied to prevent overestimation of the BZs observed in previous studies that modeled human breathing with constant inhalation rates [5]. While earlier research often simulated breathing by setting inhalation/exhalation boundary conditions on surfaces assumed to represent the nostrils or mouth, this study integrates nasal cavity and nasopharyngeal models to accurately reproduce the airflow pattern at the nostrils. The breathing profile was applied to a surface at the end of the nasopharynx (the red surface in Figure 5.1). As depicted in Figure 5.2, the unsteady breathing profile was modeled using the sine function, following the methodology proposed by Gupta et al [6]. This cycle specifies a pulmonary ventilation rate of minute volume of 7.2 L/min and a tidal volume of 0.48 L, assuming standing human body in stable condition. Figure 5.3 shows the distribution of breathing airflow over time

in the CSP with a transient breathing profile applied. The breathing profile was applied to a surface at the end of the nasopharynx. Therefore, even when the airflow rate is zero at the 2 second, it appears as though air is still exiting through the nostrils.

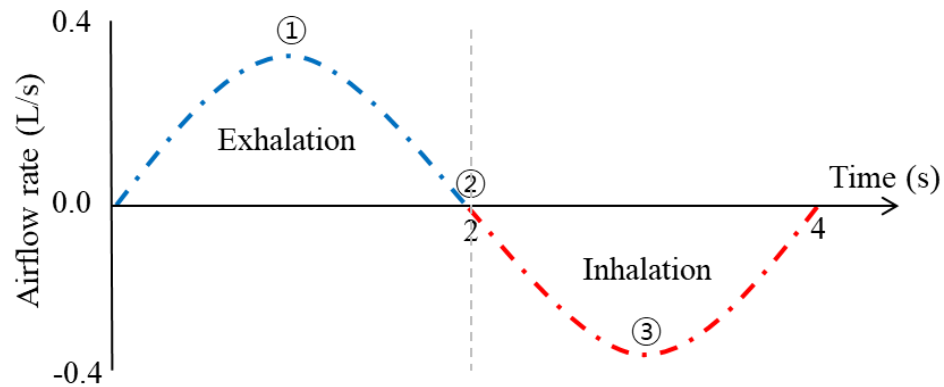


Figure 5.2. Temporal change of breathing flow rate

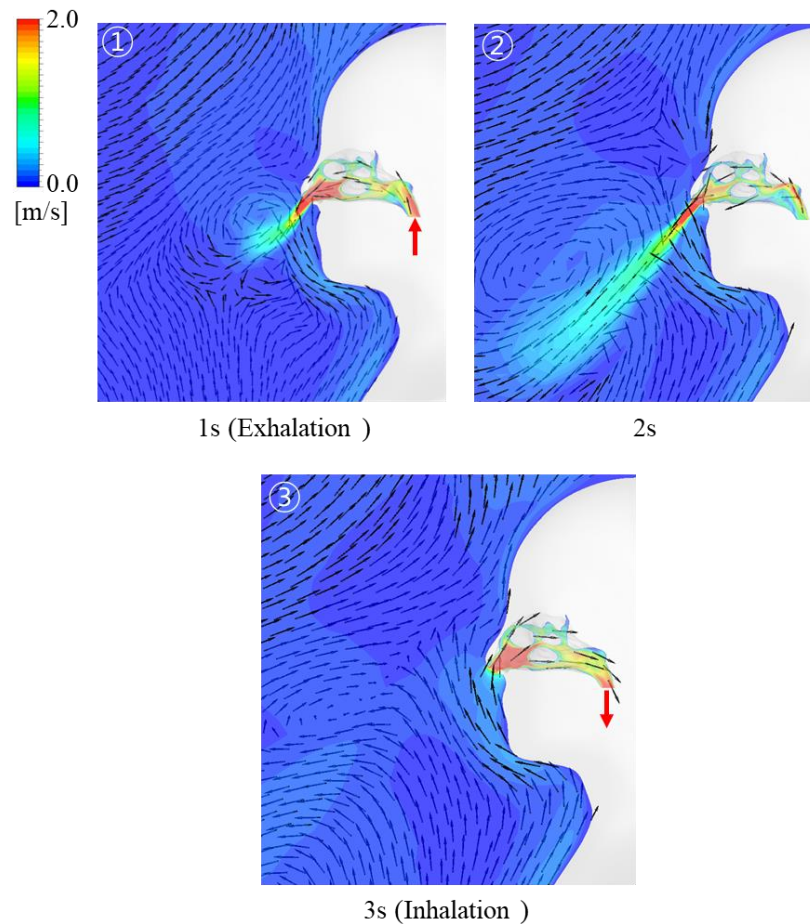


Figure 5.3. Time-dependent change of airflow pattern under transient breathing model.

5.4. CFD analysis and validation

In this chapter, I will describe the CFD domain used for validating the numerical breathing model and evaluating the effects of micro-movements, as discussed in Chapter 6. The analysis domain and boundary conditions were established based on conditions from a previous study that confirmed the prediction accuracy of flow patterns near the CSP, using a chamber experiment with a thermal manikin [7, 8]. The cubic domain was modeled as a simple room measuring $3.5 \times 3.0 \times 2.5$ m, with the CSP positioned at the center. In this study, we assumed displacement ventilation, a typical ventilation scheme widely used in residential and industrial spaces. The boundary conditions and layout of inflow/outflow boundaries to replicate displacement ventilation are illustrated in Figure 5.4. Airflow at 21.8°C and 50% relative humidity enters at a speed of 0.182 m/s. To maximize the prediction accuracy of heat and contaminant transfer on the CSP surface, an intricate grid design around the CSP is crucial. Recommended analytical approaches from a previous study, including grid design for viscous sublayer analysis, were implemented to ensure the quality of the CFD analysis utilizing the CSP [8]. Figure 5.5 depicts the grid design around the CSP and inside the nasal cavity. The surfaces of the nasal cavity and CSP were divided into polyhedral meshes measuring 0.5 mm and 3 mm, respectively, and the maximum growth rate of the volume mesh was set to 1.08. Consequently, 10 layers of boundary cells with approximately 0.2 mm thickness were applied to the CSP surfaces. As a result, the domain incorporating the CSP was discretized with 4.2 million polyhedral cells, and the dimensionless wall distance (y^{+}) remained below 1.0 over the CSP surface and nasal cavity model.

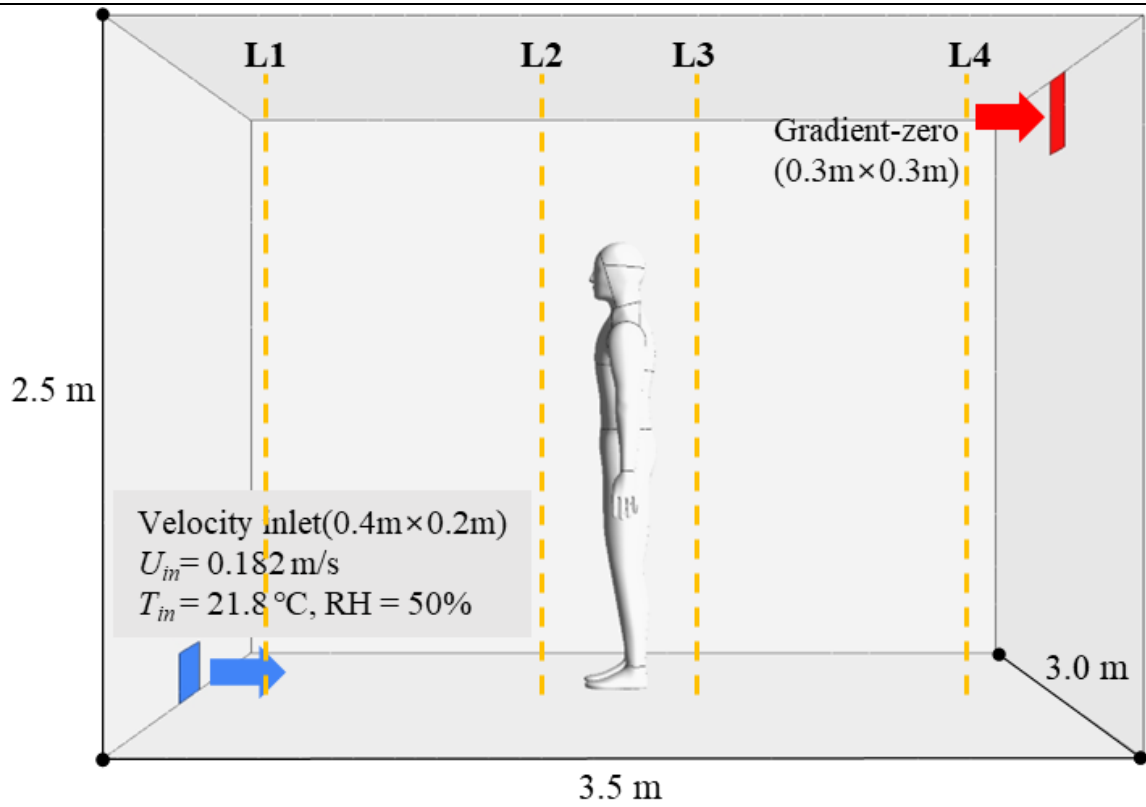


Figure 5.4. Analysis domain and boundary conditions for inlet and outlet

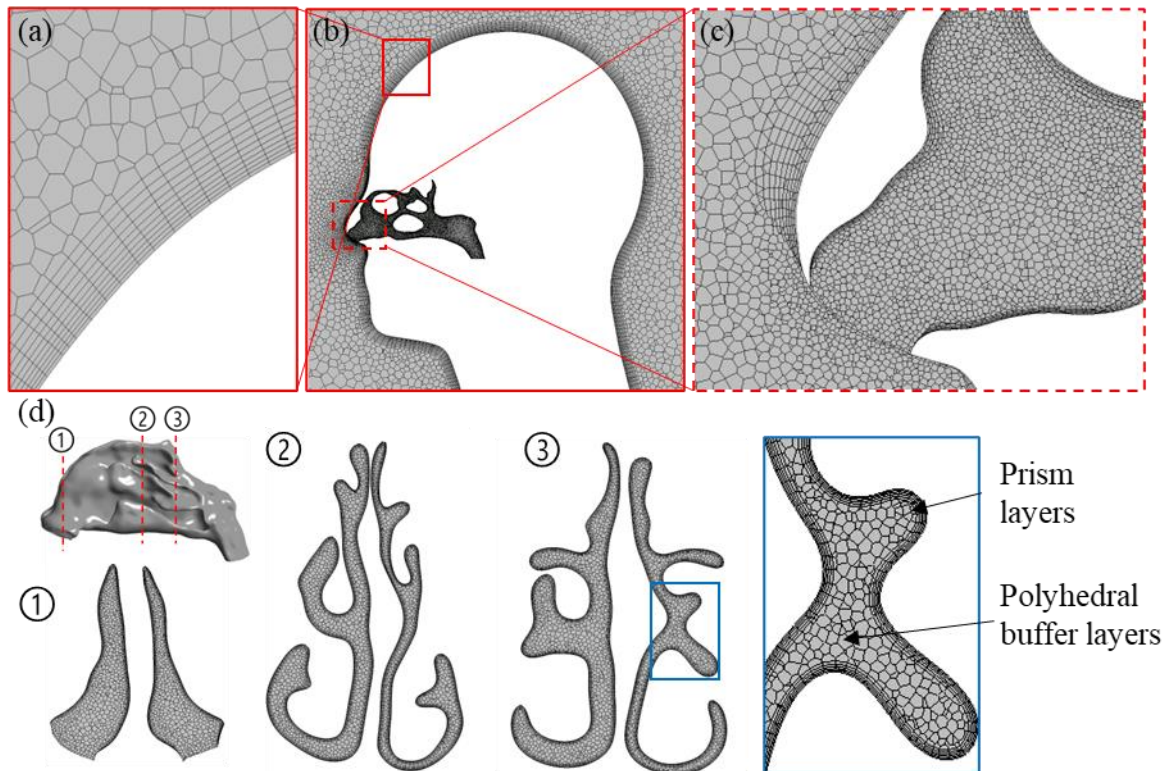


Figure 5.5. Grid detail for CSP and nasal cavity: (a) 10 boundary layers for the CSP surface, (b) local mesh around the breathing zone, (c) around the nostrils, and (d) coronal cross-sections at different locations along the nasal airway

Due to limitations in simultaneously replicating the conditions of our CFD analysis through experiments, we endeavored to ensure reliability by comparing our results with previous experimental measurements for indoor airflow and the nasal cavity, respectively. In Figure 5.6 (a), we compared our CFD analysis results with velocity measurements reported by Nielsen et al. [7], which were obtained from experiments using a thermal manikin in an environmental chamber. Since the experiment did not utilize a breathing model, the compared data were also obtained from analysis results that excluded breathing. The velocity distributions at locations L1 to L4, which are indicated in Figure 5.4, showed good agreement between the experimental and CFD results. Our numerical airway model was validated through a comparison of airflow velocities obtained from a physical model based on the same CT data in previous studies [3, 4]. However, as this study incorporated only the nasal cavity part from the airway model, further validation was conducted. Figure 5.6 (b) compares the velocity measurements taken from our numerical nasal cavity model with those from Hahn et al.'s physical model, which was based on CT data from a healthy male [9]. This analysis was independently conducted, maintaining an inhalation flow rate consistent with the experimental conditions at 10.8 L/min. Each velocity profile was normalized by the maximum velocity at the nostril. Despite variations in the nasal cavity model used, similar velocity distributions were observed at comparable positions of identical widths. Especially, it accurately reproduced the phenomenon where the velocity increases as it moves away from the wall. The above results demonstrate that sufficient prediction accuracy for both global/local climate analysis around the human body and respiratory analysis using the numerical nasal cavity can be achieved based on the indoor CFD analysis methodology using the CSP introduced in this study.

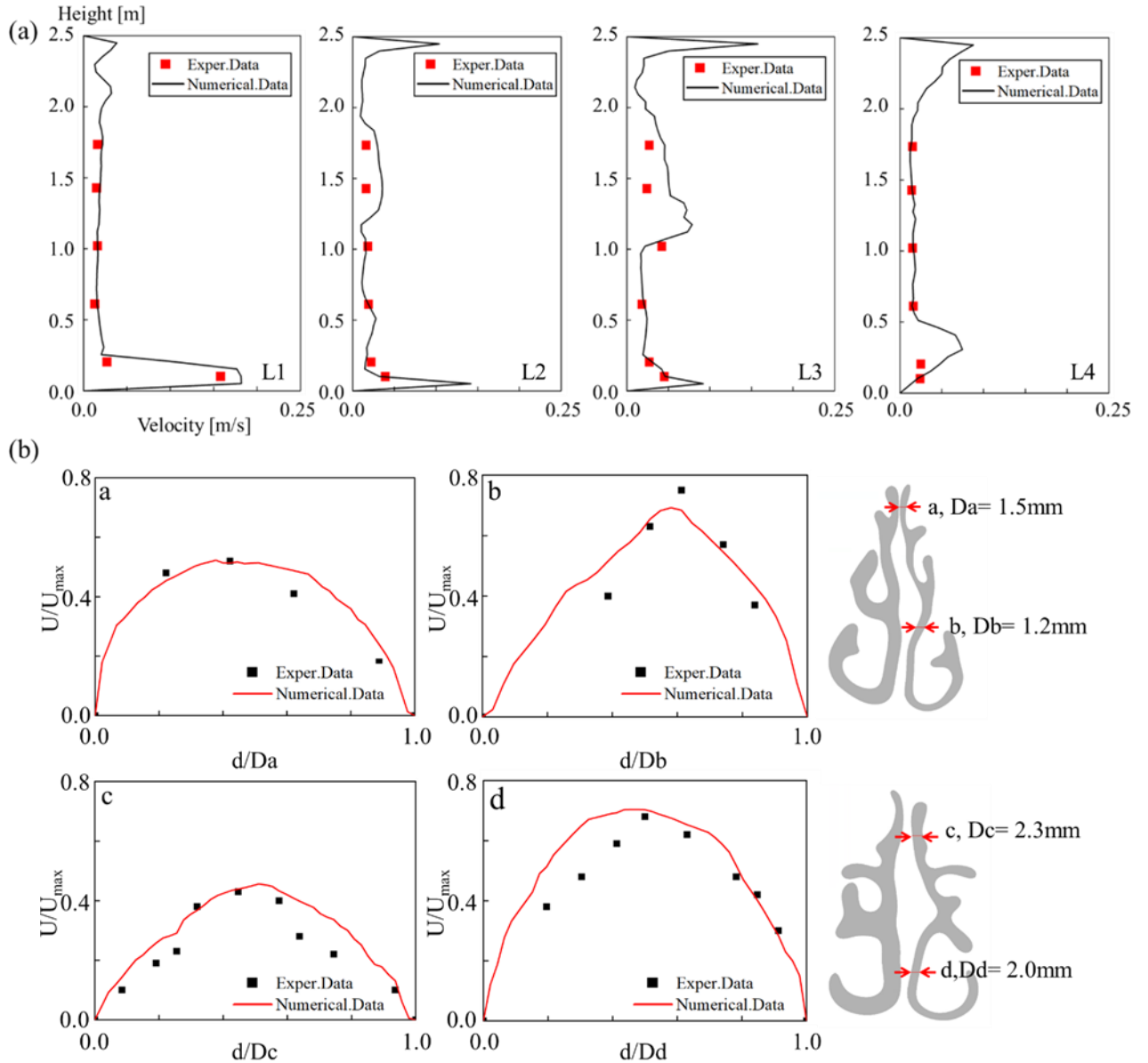


Figure 5.6. Comparison of CFD analysis-based velocity profiles with experimental measurements: (a) for lines L1, L2, L3, and L4 [Nielson et al., 2003], see Figure 5.4; (b) cross-sectional velocity profiles at slices 2 (a and b) and 3 (c and d) [Hahn et al., 1993], see Figure 5.5(d)

5.5. Scale for Ventilation Efficiency (SVE) 4 and 5 for analyzing the BZ

In this study, SVE4 and SVE5, as proposed by Kato and Murakami [10, 11] were employed to estimate the time-dependent transition of the BZs. SVE4 and SVE5 represent the contribution ratios of the supply and exhaust openings for indoor ventilation efficiency, respectively. Specifically, the nostrils were considered as the supply and exhaust openings in this analysis.

Thus, to evaluate the distribution of exhaled air, SVE4 was applied in the following manner:

$$\text{SVE4} = \frac{C(x, n)}{C_0(n)} \quad (5.1)$$

$C(x, n)$ represents the concentration of the tracer gas at point x , and $C_0(n)$ denotes the concentration at the supply opening (from the nasal cavity).

$$\text{SVE5} = \frac{C(x, m)}{C_0(m)} \quad (5.2)$$

Conversely, SVE5 was used to assess the distribution probability of reaching to nostrils and calculated by Equation (5.2), which is based on the distribution of tracer gas obtained through reverse simulation. In this equation, $C(x, m)$ represents the concentration of the inhaled air at point x , $C_0(m)$ denotes the concentration at the n^{th} exhaust opening (in the nasal cavity). When initializing the calculations, the non-dimensional tracer gas concentrations were set to 0 and 1 in the indoor area and the nasal cavity, respectively. This methodology contrasts with the exhaled air analysis, where the natural process from the supply opening to the room aligns with forward-time tracking. The reversing simulation tracks air backward from the exhaust openings to the room to understand how air at a specific point reaches these openings [2, 5, 12].

To predict the BZ distribution, the passive scalar was treated as a tracer gas, utilizing the scalar transport equation described in Chapter 2.5 to predict tracer gas distribution. Distinguishing between inhaled and exhaled air is crucial for understanding and managing various aspects of airflow. For example, exhaled air is primarily analyzed to study the dispersion of pathogens and viruses, information crucial for designing and improving ventilation systems in public spaces or hospitals to reduce infection risk. Conversely, inhaled air relates directly to the quality of air that individuals breathe, critical for evaluating the efficiency of personal ventilation systems and

infection prevention measures. By assessing air quality in specific environments, targeted improvements can be made that significantly enhance personal health and safety. Thus, differentiating and understanding these two types of BZs are vital in fields such as architecture design, environmental control, public health protection, and personal health management.

REFERENCES

- [1] J. Huang, P. Jones, A. Zhang, S.S. Hou, J. Hang, J.D. Spengler, Outdoor Airborne Transmission of Coronavirus Among Apartments in High-Density Cities, *Front. Built Environ.* 7 (2021) 666923.
- [2] I. Abouelhamd, K. Kuga, S.-J. Yoo, K. Ito, Identification of probabilistic size of breathing zone during single inhalation phase in semi-outdoor environmental scenarios, *Building and Environment* 243 (2023) 110672.
- [3] N.L. Phuong, K. Ito, Investigation of flow pattern in upper human airway including oral and nasal inhalation by PIV and CFD, *Building and Environment* 94 (2015) 504–515.
- [4] N.L. Phuong, M. Yamashita, S.-J. Yoo, K. Ito, Prediction of convective heat transfer coefficient of human upper and lower airway surfaces in steady and unsteady breathing conditions, *Building and Environment* 100 (2016) 172–185.
- [5] K. Kuga, P. Wargocki, K. Ito, Breathing zone and exhaled air re-inhalation rate under transient conditions assessed with a computer-simulated person, *Indoor Air* 32 (2022).
- [6] J.K. Gupta, C.-H. Lin, Q. Chen, Characterizing exhaled airflow from breathing and talking, *Indoor Air* 20 (2010) 31–39.
- [7] P.V. Nielsen, S. Murakami, S. Kato, C. Topp, J.-H. Yang, Benchmark tests for a computer simulated person, Aalborg University, *Indoor Environmental Engineering*. (2003).
- [8] S. Yoo, K. Ito, Validation, verification, and quality control of computational fluid dynamics analysis for indoor environments using a computer-simulated person with respiratory tract, *Japan Architectural Review*. 5 (2022) 714–727.
- [9] I. Hahn, P.W. Scherer, M.M. Mozell, Velocity profiles measured for airflow through a large-scale model of the human nasal cavity, *Journal of Applied Physiology* 75 (1993) 2273–2287.
- [10] S. Kato, S. Murakami, H. Kobayashi, New scales for evaluating ventilation efficiency as affected by supply and exhaust openings based on spatial distribution of contaminant., (1992).
- [11] S. Murakami, New scales for ventilation efficiency and their application based on numerical simulation of room Airflow, *J. Wind Eng.* 52 (1992) 132–148.
- [12] I. Abouelhamd, K. Kuga, S.-J. Yoo, K. Ito, Effect of Crowd Density, Wind Direction, and Air Temperature on the Configuration of Individual Human Breathing Zones in a Semi-Outdoor Environment, *SSRN*, (2023).

CHAPTER 6

Micro-movement

6.1. Introduction

In this chapter, we discuss micro-movements resulting from posture sway, a characteristic of human physiology, and introduces the single-mass inverted pendulum model (IPM) developed to simulate these movements. Additionally, the chapter presents the results of CFD analysis that investigates how these micro-movements influence the formation of the CSP's BZ and affect the general indoor climate.

6.2. Review of previous research on human movement

The impact of physical activities such as walking or running on IEQ and the microclimate surrounding the human body has been extensively studied. These activities alter the microclimate around the body, significantly affecting heat transfer and contaminant concentrations in the breathing zone [1-3]. In addition to these active movements, the human body also experiences constant micromovements while standing or sitting [4]. These are often manifested as postural sway, resulting from variations in torque at the ankles, knees, and hip joints, regulated by neural control pathways. Notably, the degree of postural sway varies with age and health conditions, underscoring its dependency on individual physiological states [5]. These micro-movements are small but persistent, suggesting a potential lasting impact on indoor environments, though they have largely been overlooked in IEQ research. Yoo et al. modeled a young, healthy adult under typical indoor environmental conditions and found that the reproduction of micro-movements had a negligible effect on convective heat and mass transfer. However, they suggested the impact

might vary depending on the subject's conditions [4]. Despite evidence that physical activities significantly influence contaminant concentrations in occupant's BZ, previous studies defining BZ have not considered the human body's micro-movements. Therefore, there is a clear need to evaluate the effects of micro-movements in defining the BZs.

Against this background, we investigated the impact of micro-movements on the microclimate surrounding the human body, and on the size and shape of breathing zones (BZs) using a standing CSP integrated with a numerical nasal cavity model, as detailed in the chapter.

6.3. CFD analysis for the effect of human micro-movement on BZ formation

To assess the impact of the CSP's micro-movements on the BZ and the indoor environment, the CFD domain and boundary conditions validated in Chapter 5 were used. The CFD simulation enabled the calculation of airflow, temperature, and passive scalar concentration distributions within the human respiratory system and indoor space under transient breathing conditions. The Reynolds-Averaged Navier-Stokes (RANS) equations were solved using the shear stress transport (SST) $k-\omega$ turbulence model, which has shown good agreement with experimental measurements in convective heat transfer analysis around the CSP [6, 7]. By employing a $k-\omega$ formulation in the near-wall viscous sublayer and transitioning to a $k-\epsilon$ behavior in the free-stream regions, this model provides a hybrid approach ideally suited for predicting flow near walls and far-field regions. Table 6.1 details the numerical and boundary conditions for the CFD. Pressure-velocity coupling was addressed using the semi-implicit method for pressure-linked equations (SIMPLE) algorithm. Convergence was deemed achieved when residuals stabilized and met the specified criteria of 10^{-5} in the transient-state iterative calculations.

Table 6.1. Boundary conditions for CFD calculation

Algorithm	SIMPLE (Unsteady calculation)
Turbulence model	SST $k-\omega$ model
Discretization scheme	Second-order upwind scheme for convection
Inflow boundary (Room)	Velocity: 0.182 m/s, Turbulent intensity = 10% Temperature: 21.8 °C, Relative Humidity: 50%
Inflow boundary (Airway)	Transient breathing profile, Flow rate = 7.2 L/min 1 cycle in 4s (Inhalation: 2s & Exhalation: 2s)
Outflow boundary	Gradient-zero
Wall treatment	No-slip for all wall boundaries
Wall boundary condition (Room)	Emissivity: skin = 0.95
Wall treatment (CSP)	Temperature: Calculated by Fiala thermoregulation model Humidity: Water vapor generation assuming 6 % of skin wettedness Emissivity: skin = 0.98
Wall treatment (Airway)	Temperature = 36.4 °C, and Emissivity = 0.98
Radiation model	Surface to Surface (S2S) model with view factor calculation

6.4. Reproduction of human postural sway

Humans are bipedal and inherently possess an unstable system due to the concentration of body mass in the upper body, requiring an active balance control system even when standing on both feet [8]. Given that a bipedal upright stance resembles an inverted pendulum, numerous postural control studies have employed the single-mass inverted pendulum model (IPM). In this model, the body's center of mass (COM) pivots around the ankle joint to accurately represent the body's dynamics during standing [9-12]. Despite the increasing use of multi-segment IPM modeling, which offers a detailed representation of the interplay between ankle stiffness and gravitational stiffness [13, 14], this study utilizes the single-segment IPM model. The simplicity of this model, along with its focus on the ankle joint as the pivot point, has been extensively

validated in previous research for accurately predicting the body's swaying movement. It provides a streamlined method for examining the fundamental aspects of postural control [15, 16]. In this study, we employed the single-segment IPM approach to determine the sway angular velocity and frequency for the standing CSP. As shown in Figure 6.1 (a), we applied postural sway frequencies of 1.0 Hz in the anteroposterior (AP) direction and 0.5 Hz in the mediolateral (ML) direction, with a maximum angular velocity of 4.0-5.0 °/s [4]. These postural sway frequencies and maximum angular velocities were determined targeting young and healthy adults, based on values previously reported [17-19].

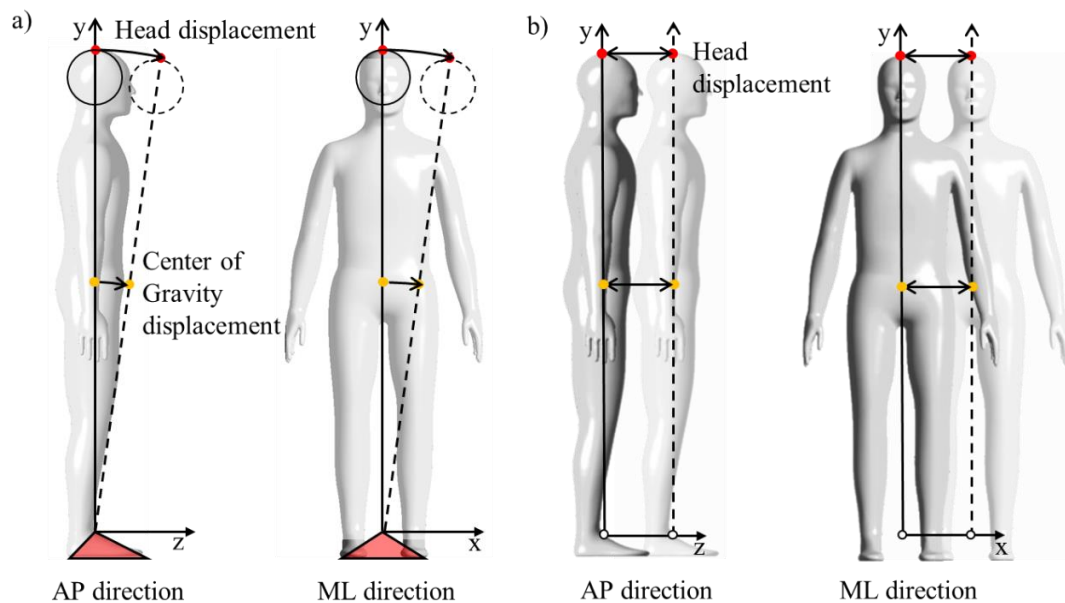


Figure 6.1. Schematic of (a) the single-segment IPM and human postural sways and (b) the linear movement in the AP and ML directions

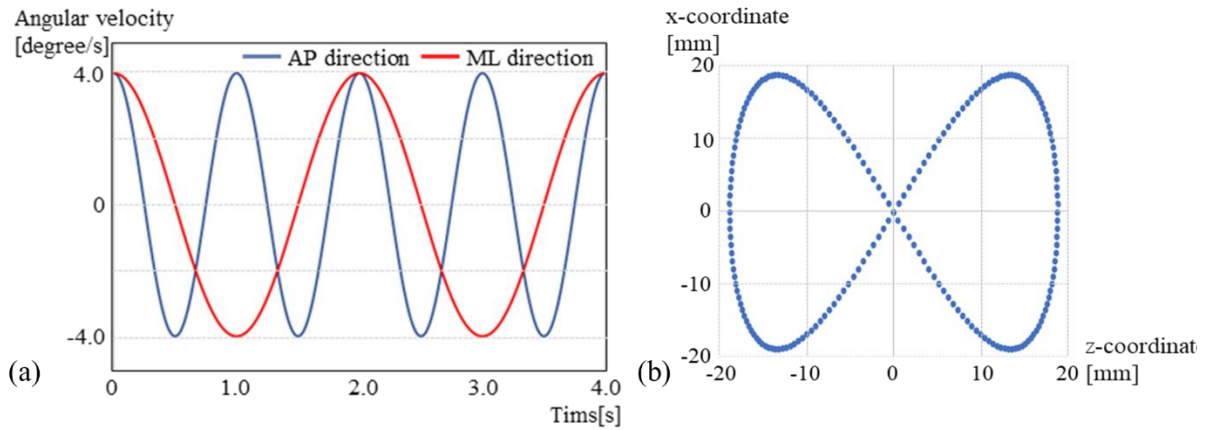


Figure 6.2. (a) Angular velocity of the CSP in the AP and ML directions (b) head displacement at each time step (for $\omega_{\max} = 4.0$ °/s)

To evaluate the variations based on the method of replicating micro-movements, we analyzed the impact on BZ formation of micro-movements using a linear movement in the AP/ML direction. As shown in Figure 6.1 (b), a linear movement with a frequency of 0.5Hz was applied in each AP and ML direction. The range of movement was set to oscillate 2cm to either side from the baseline, correlating with approximately 2cm of head displacements in IPM movement with a maximum angular velocity of 4.0 °/s, as shown in Figure 6.2. To incorporate these movements into the CSP-CFD coupled analysis, a dynamic meshing in ANSYS Fluent 2022R2 was utilized. To prevent numerical errors due to boundary mesh changes, the 10 layers of boundary meshes on the CSP surface were maintained during mesh updates.

To analyze the impact of human micro-movement on the BZ, this study established five analysis cases, as detailed in Table 6.2. These cases varied based on the presence or absence of movement in the CSP, the range of maximum angular velocity, and the type of movement implemented. Case 1, featuring no movement, was established as a control group for comparison against the other cases with movement. Cases 2 and 3 implemented a single IPM movement: Case 2 was set at a maximum angular velocity ($\omega_{\max}$) of 4°/s, and Case 3 set at 5°/s. Case 4 enabled

Table 6.2. Numerical analysis cases

Case 1	Without movement
Case 2	Angular movement, $\omega_{\max} = 4^\circ/\text{s}$
Case 3	Angular movement, $\omega_{\max} = 5^\circ/\text{s}$
Case 4	Linear movement, ML direction
Case 5	Linear movement, AP direction

the CSP to execute linear movements in the ML direction, while Case 5 was designed for linear movement in the AL direction. In Cases 4 and 5, the displacement of the CSP was configured to oscillate up to 2cm from the initial reference point, consistent with the head movement observed in Case 2.

6.5. CFD analysis results and discussion

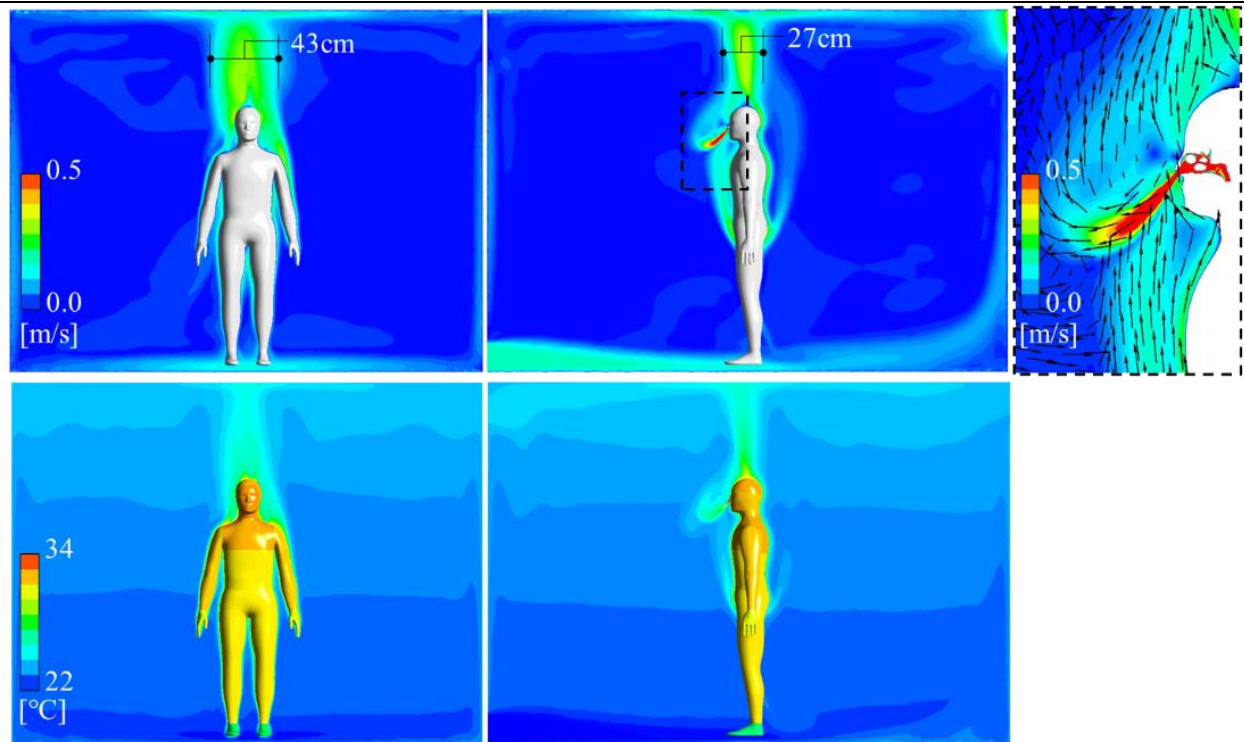
6.5.1. Airflow and temperature distribution

Figure 6.3 illustrates the results of the airflow vector, scalar velocity, and temperature distribution analysis around the CSP for each case at the end of exhalation. This study utilized displacement ventilation, characterized by buoyancy-driven, low-speed air inflow that avoids creating rapid temperature fluctuations and strong airflows within the environment. The inflow air, passing between the legs of the CSP, ascended along the wall opposite the inlet at a diminishing speed, minimally affecting the CSP or the surrounding airflow. Consequently, in all cases, there was a clear indication of rising airflows around the CSP, approximately 0.2-0.3 m/s, occurred by metabolic heat generation of human body, and thermal stratification in the whole domain. Moreover, the use of the multi-segment thermoregulation model allowed for detailed

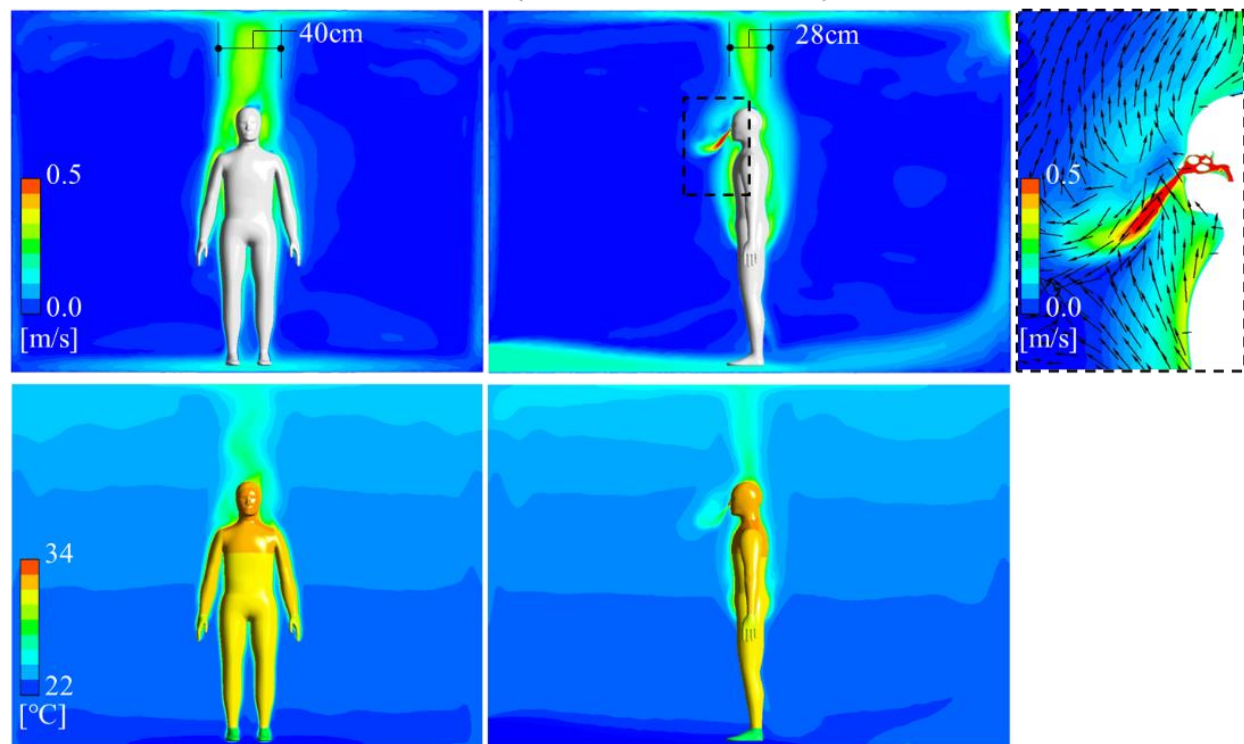
observation of distinct temperature distributions across different segments of the CSP, particularly noting cooler temperatures in extremities such as the hands and feet, similar to observations from previous experiments with the ANDI thermal manikin under comparable conditions [20].

The study quantitatively assessed the impact of micro-movements on the thermal plume generated by the CSP's heat. Measurement of airflow distribution with velocities exceeding 0.1 m/s indicated that the maximum width of the thermal plume varied by no more than 3 cm, regardless of micro-movement presence. Specifically, the maximum widths were 43cm in the front and 27cm on the side for Case 1. For Case 2, the widths measured 40cm (front) and 28cm (side); for Case 3, they were 41cm (front) and 27cm (side); Case 4 exhibited widths of 44cm (front) and 28cm (side); and Case 5 showed widths of 42cm (front) and 30cm (side). Additionally, there was almost no variation in the height distribution of thermal plume.

These findings indicate that the micro-movements of the CSP have a relatively limited effect on the maximum width of the thermal plume, suggesting that such movements do not significantly alter macroscopic indoor airflow and temperature distributions. However, the influence of micro-movements on the airflow patterns near the CSP was notable. Specifically, a notably wider distribution of rising airflows was observed at the CSP's rear in cases with movement.

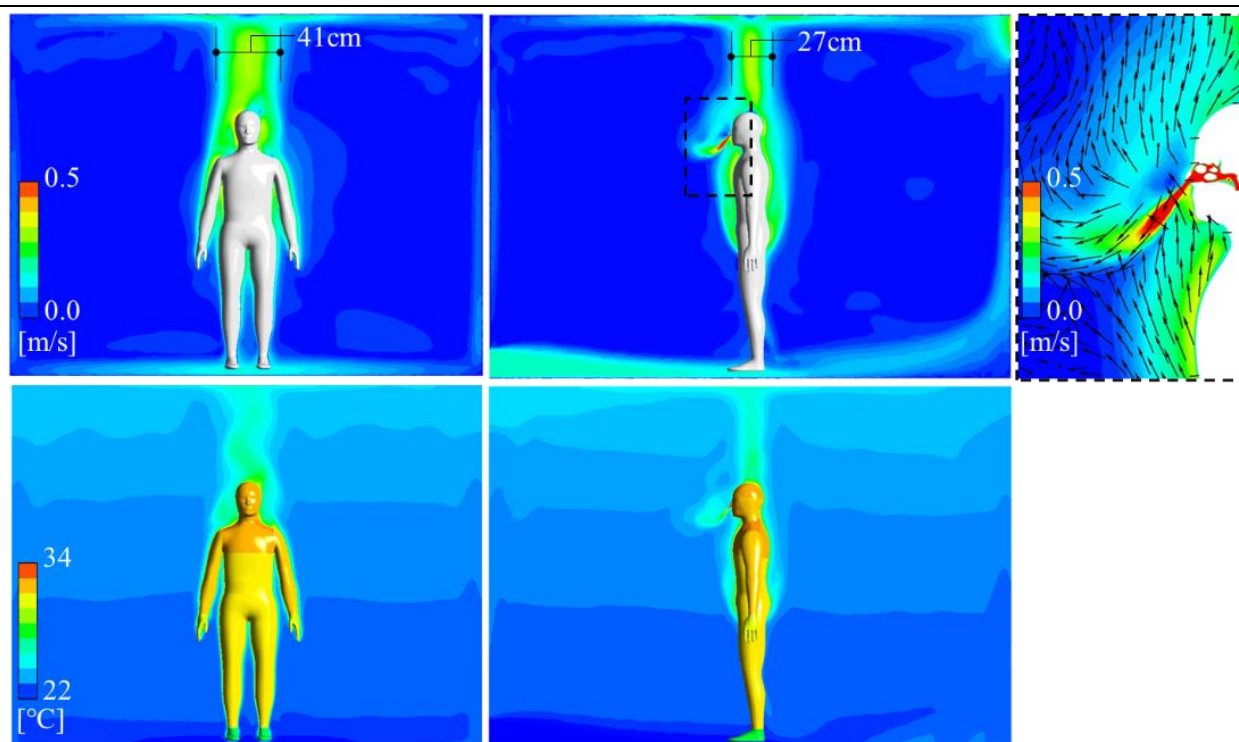


Case 1 (Without micro-movement)

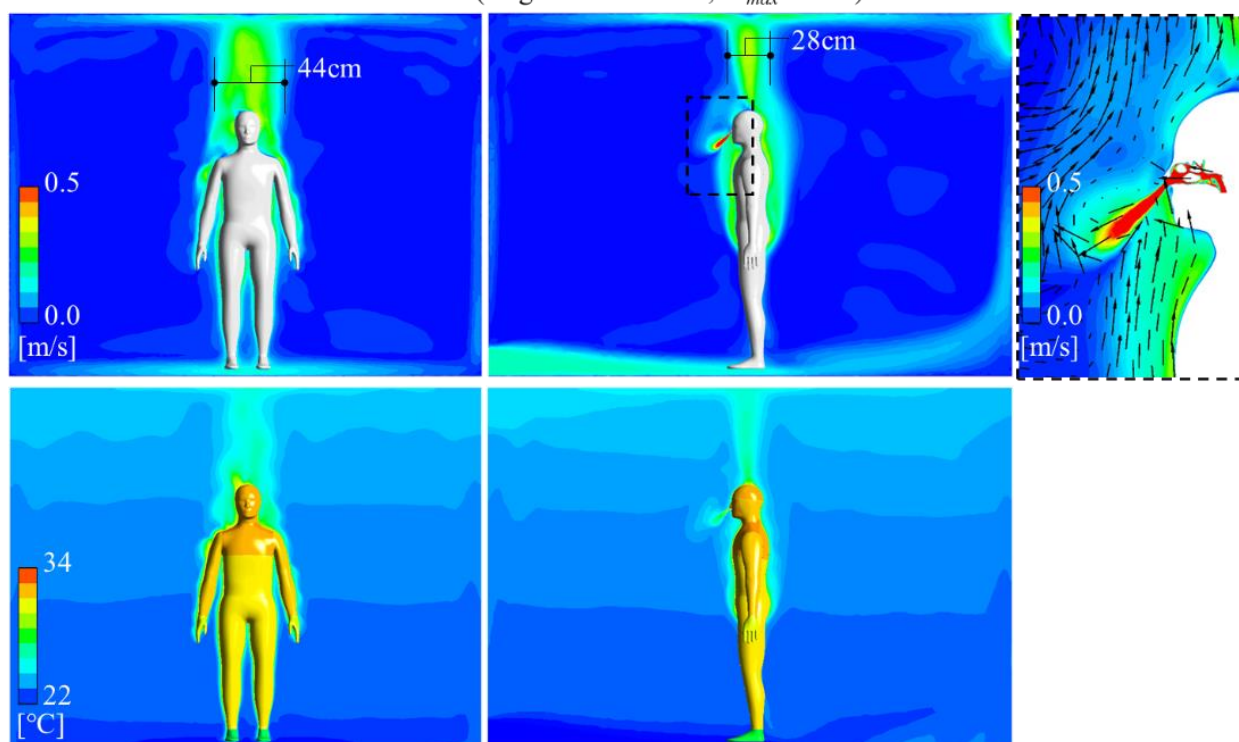


Case 2 (Angular movement, $\omega_{max} = 4^\circ/s$)

Figure 6.3. Airflow pattern and temperature distribution around CSP

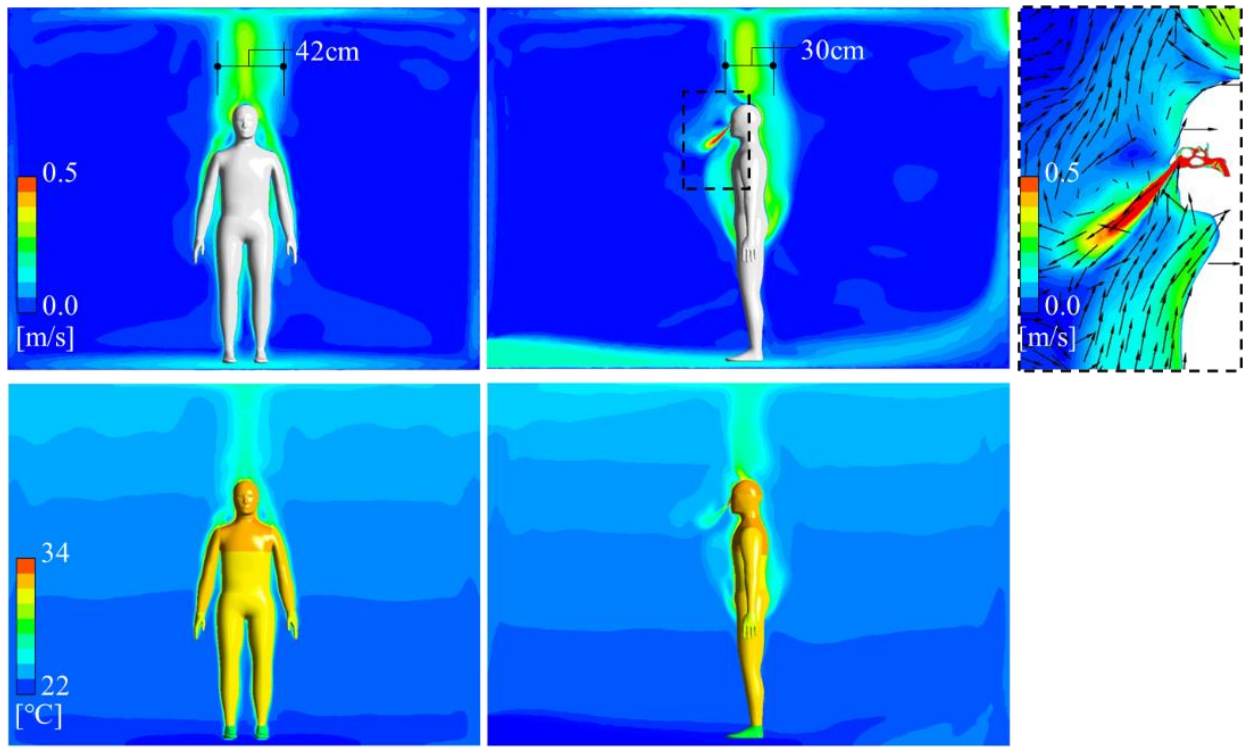


Case 3 (Angular movement, $\omega_{max} = 5^\circ/s$)



Case 4 (Linear movement, ML direction)

Figure 6.3. (Continued)



Case 5 (Linear movement, AP direction)

Figure 6.3. (Continued)

In Cases 2 and 3, which involved angular movements, a slight increase in the speed of rising airflows was observed at the abdomen and chest levels of the CSP, though this increase resulted in reduced airflow stability. Similarly, there was an increase in the speed of was also observed in Cases 4 and 5. However, in Case 5, it was observed that the direction of the rising airflow flowed towards the CSP. This alteration in the airflow around the front of the CSP suggests potential impacts on the breathing zones (BZs). It is considered that this variation in the diffusion of updraft could significantly influence the distribution within the human breathing zone.

6.5.2. Convective heat transfer coefficient on the CSP surface

To evaluate the impact of micro-movements on heat transfer, the convective heat transfer coefficients (CHTC) on the surface of the CSP were estimated for each case. Figure 6.4 illustrates the distribution of CHTC across the CSP's surface in each case, while the area-weighted averages of CHTC are summarized in Table 6.3. Despite the incorporation of movement and an increase in maximum angular velocity, the rises in the convective heat transfer coefficients for Cases 2 and 3 were modest, at only 2.51% and 3.63% respectively, compared to Case 1, suggesting a negligible impact. In contrast, Cases 4 and 5, which involved linear movements, exhibited higher increases in CHTC. This was attributed to more extensive body movements of the CSP, especially noticeable in the limbs as opposed to the torso. The increases in these cases were 5.31% and 11.17%, respectively, compared to Case 1.

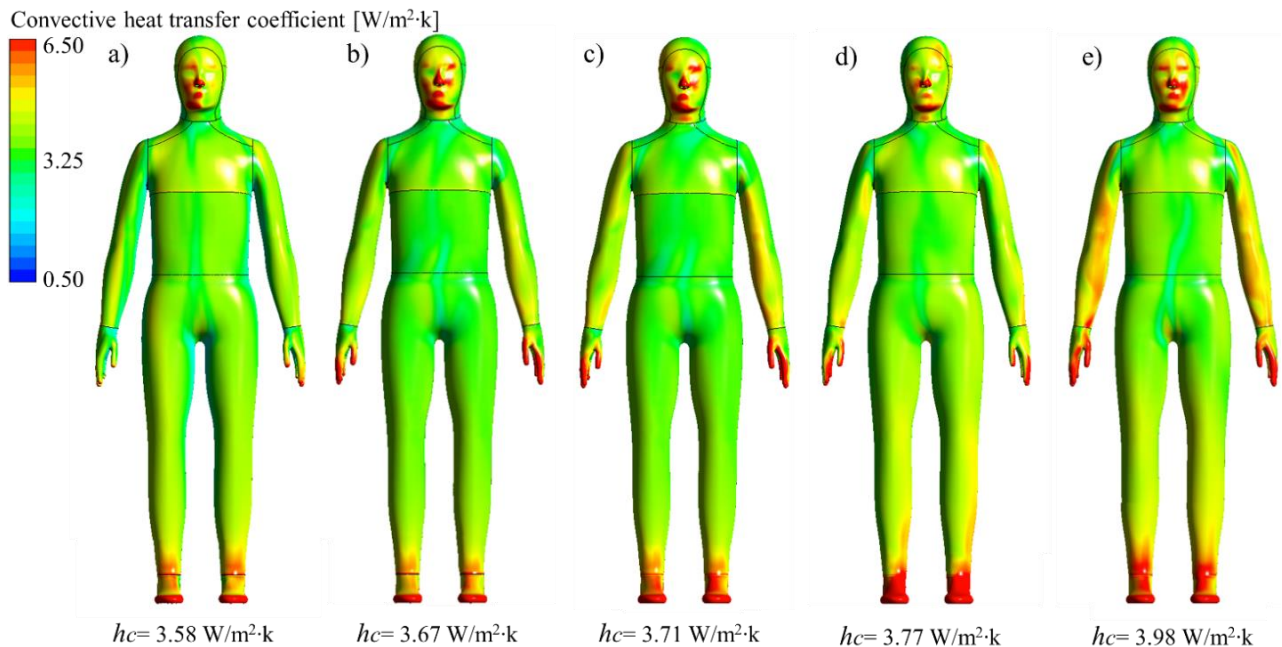


Figure 6.4. Distribution of convective heat transfer coefficient on the CSP surface; a) Case 1, b) Case 2, c) Case 3, d) Case 4, and e) Case 5.

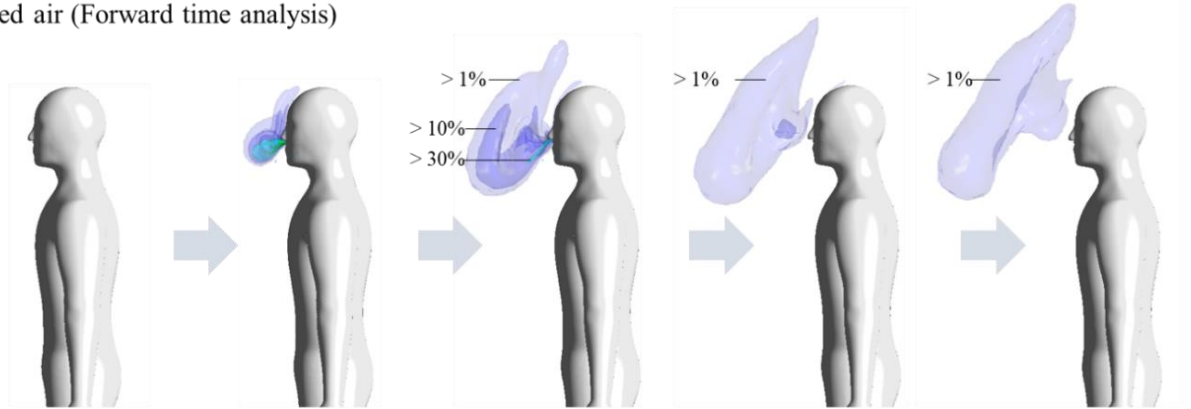
Table 6.3. Comparison of the averaged convective heat transfer coefficient on the CSP surface according to the case

	Averaged CHTC (W/m ² ·K)	Increase compared to Case 1 (%)
Case 1 (Stationary)	3.58	-
Case 2 (Angular movement, $\omega_{\max}=4.0^{\circ}/s$)	3.67	2.51
Case 3 (Angular movement, $\omega_{\max}=5.0^{\circ}/s$)	3.71	3.63
Case 4 (Linear movement, ML)	3.77	5.31
Case 5 (Linear movement, AP)	3.98	11.17

6.5.3. Breathing zone

Breathing Zones (BZs) were identified using SVE4 and SVE5, as introduced in Chapter 5, and Figure 6.5 illustrates the time-series formation process of exhaled and inhaled air in Case 1. The dispersion of air exhaled through the nostrils into the surrounding environment is analyzed using SVE4 calculations, which are based on forward-time flow field analysis. Initially, for up to 2 s, the exhaled air is carried by both the exhalation jet and the thermal plume. Beyond 2 s, it is observed diffusing away from the CSP via rising airflow. Conversely, the tracing origin of air just before being inhaled through the nostrils, referred to as inhaled air, is analyzed using SVE5 calculations based on inverse simulation. This inhaled air is initially distributed near the abdomen and chest of the CSP at 0 s and is carried upward by the rising airflow by the thermal plume. At 2 s, coinciding with the transition from exhalation to inhalation, the exhaled air is distributed from the nostrils down to the abdomen, facilitating a comparison of inhaled air distribution across different cases.

a) Exhaled air (Forward time analysis)



b) Inhaled air (Reverse time analysis)

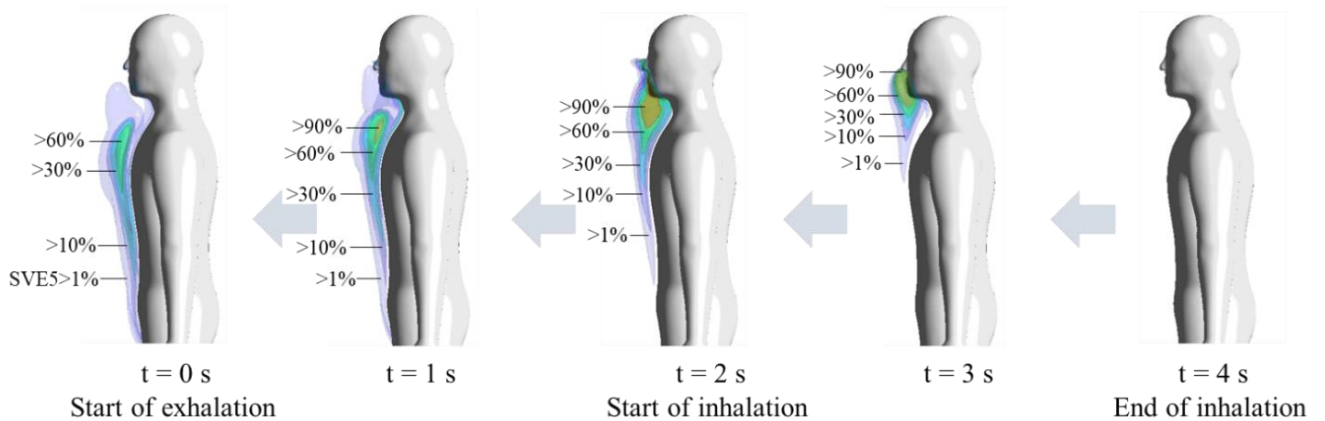
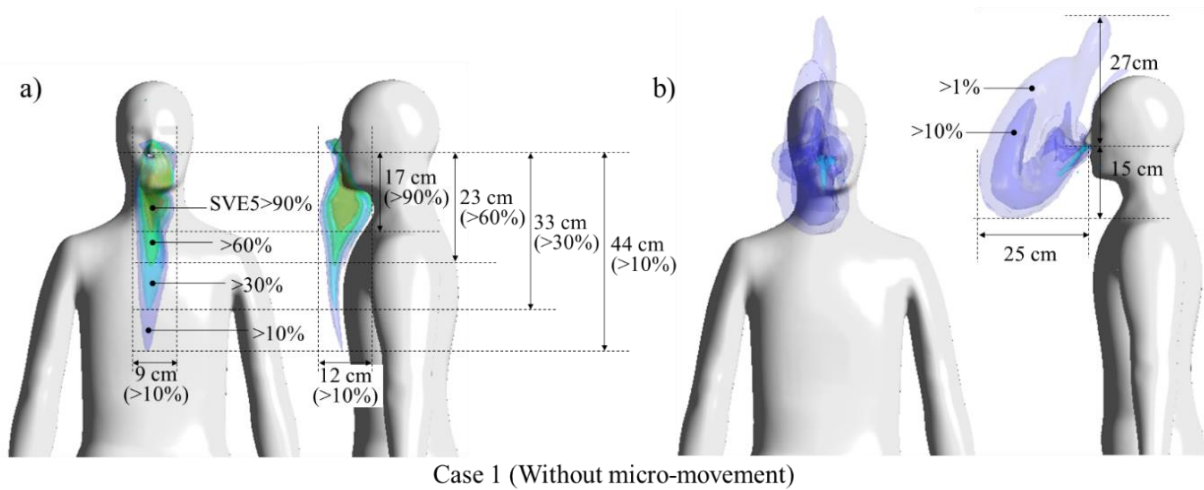


Figure 6.5. The time-series formation process of the breathing zone:(a) Exhaled air (b) Inhaled air



Case 1 (Without micro-movement)

Figure 6.6. Breathing zone of CSP without micro-movement: (a)Inhaled air distribution is visualized using SVE5, (b) Exhaled air distribution

Figure 6.6 (a) illustrates the distribution of SVE5 exceeding 10% in Case 1, utilizing a CSP without micro-movements. The higher SVE5 value indicates higher probability of air being inhaled through the nostrils. It was observed that the SVE5 distribution originated downward

from the nose due to the rising airflow at the front of the CSP. The inhaled air extended from the nasal opening to the epigastrium, measuring 44cm vertically and 9cm horizontally. Moreover, areas with a high probability of inhalation, exceeding 90%, showed a concentration of inhaled air extending from the nose to below the chin.

Additionally, exhaled air is initially discharged from the CSP's nostrils at approximately a 45° downward angle. However, as depicted in Figure 6.6 (b), after descending about 15 cm, it rises due to the thermal plume surrounding the CSP.

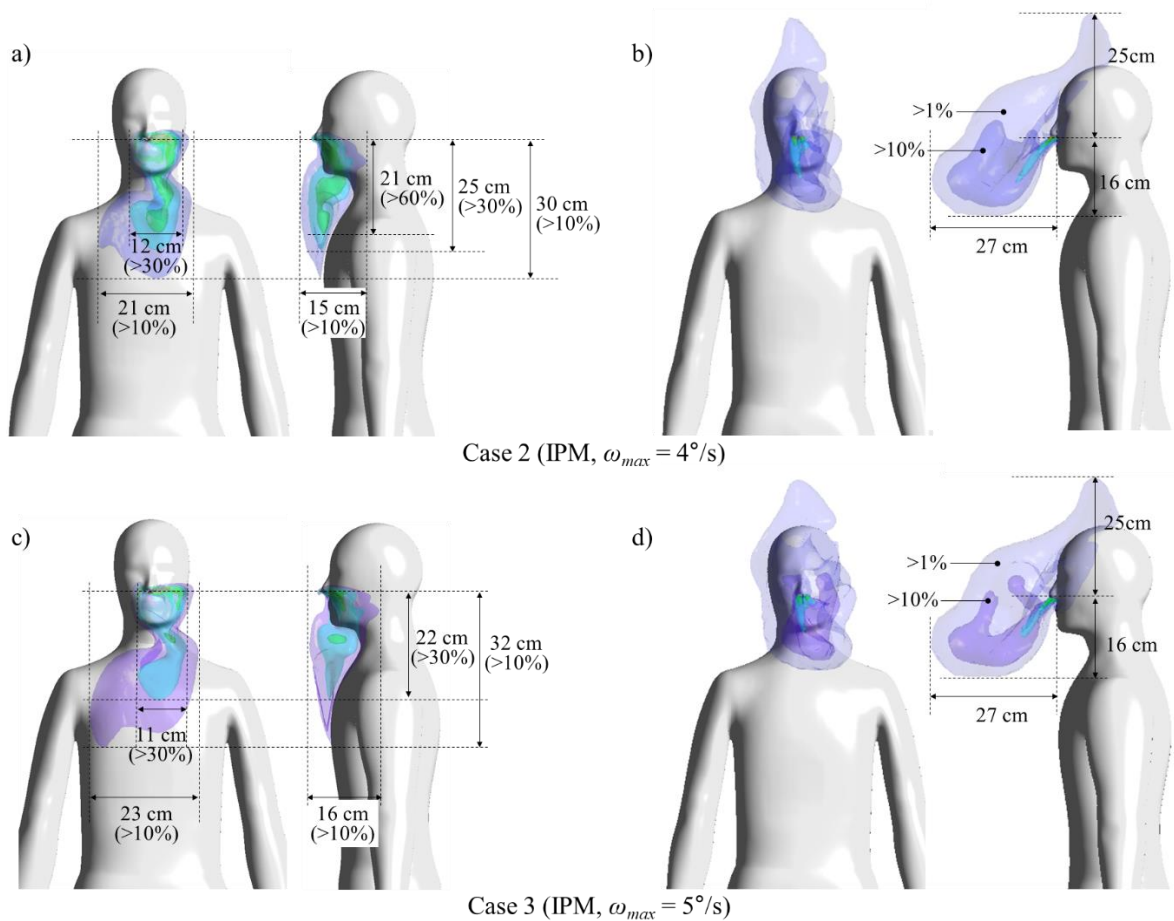


Figure 6.7. Breathing zone of CSP with angular movement: (a) Inhaled air distribution of Case 2, (b) Exhaled air distribution of Case 2, (c) Inhaled air distribution of Case 3, (d) Exhaled air distribution of Case 3

Figure 6.7 illustrates the BZ for Cases 2 and 3, in which angular movements were applied to the CSP to simulate human postural sway. In these cases, the continuous movement of the nostrils led to a reduction in the vertical lengths of the SVE5 (inhaled air) distribution: 30 cm in Case 2 and 34 cm in Case 3, both shorter than those observed in Case 1. Conversely, the horizontal lengths increased to 21 cm in Case 2 and 24 cm in Case 3. Unlike in Case 1, there was almost no distribution of SVE5 exceeding 90% except near the nostrils. In Case 3, which had a maximum angular velocity 1°/s higher than Case 2, resulting in approximately 1.0 cm wider head displacement, an increase in the range of >10% concentration distribution was observed both vertically and horizontally, while the range of >60% concentration decreased. In Cases 2 and 3, after being expelled at approximately a 45° downward angle, the exhaled air exhibited a rising pattern due to the thermal plume. Although Case 3 involved wider movements than Case 2, leading to a narrower distribution of >10% concentration compared to Case 2, the overall distribution patterns remained similar.

In these cases, the exhaled air was distributed closer to the Computational Simulated Person (CSP) than in Case 1, with the range and size of the distributions generally consistent across both cases. The single Inverse Path Mapping (IPM) movement of the CSP had a more pronounced effect on the distribution of inhaled air than on exhaled air. This movement increased the horizontal extent and decreased the vertical reach of the inhaled air distribution, with the magnitude of these distribution changes exceeding the level of head displacement.

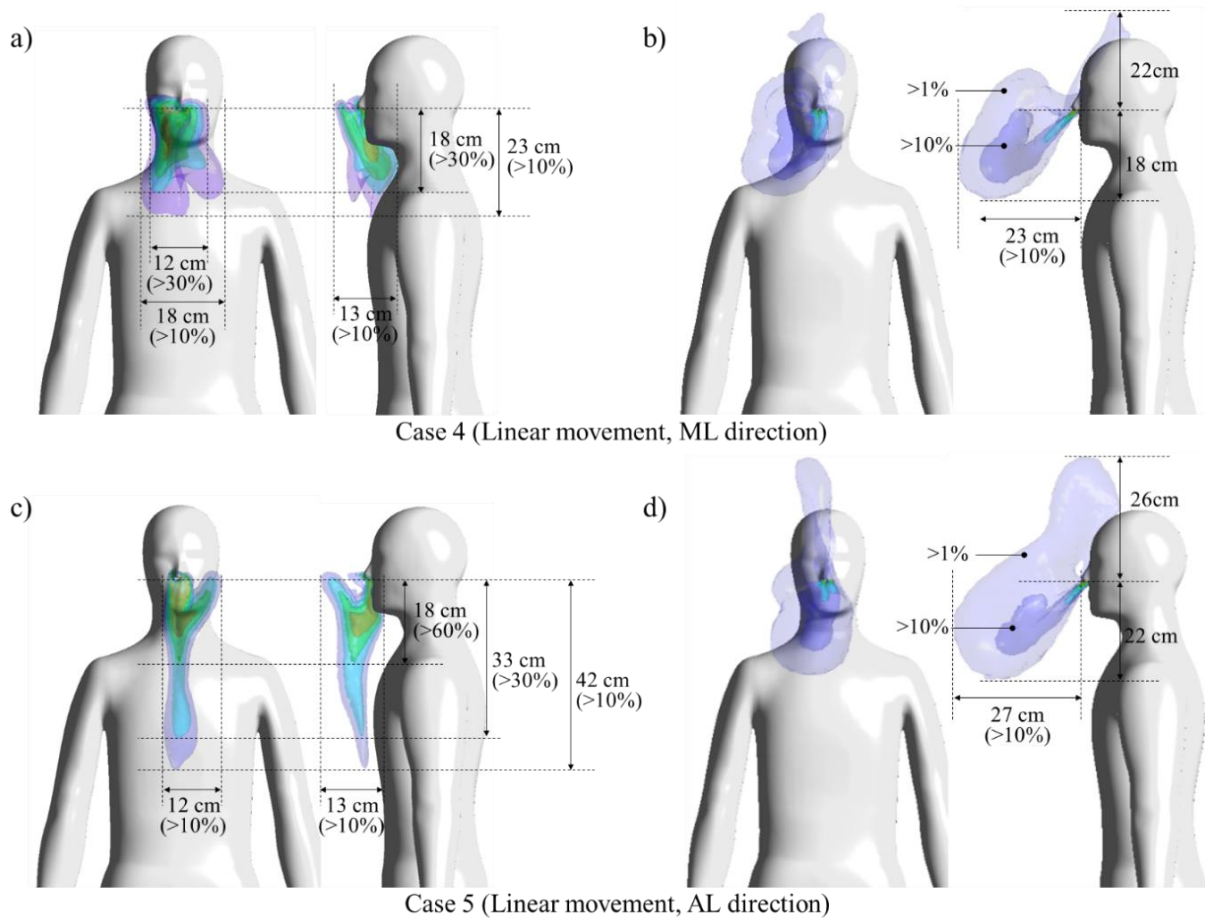


Figure 6.8. Breathing zone of CSP with linear micro-movement: (a) Inhaled air distribution of Case 4, (b) Exhaled air distribution of Case 4, (c) Inhaled air distribution of Case 5, (d) Exhaled air distribution of Case 5

Figure 6.8 presents the BZ distribution results for Case 4 and Case 5, where linear movements were applied to assess their impact on BZ formation. In Case 4 with linear movement in the ML direction, the inhaled air distribution with a concentration $>10\%$ was 23 cm vertically (a reduction of 7 cm from Case 1) and 18 cm horizontally (an increase of 9 cm from Case 1). The shape of this distribution was relatively similar that observed in Case 2, which involved single IPM movement. However, a distinct difference in Case 4 was the presence of a concentration distribution exceeding 90%, which was almost non-existent in Case 2. Conversely, in Case 5, where the linear movement was applied in the AL direction, the distribution of inhaled air with a concentration greater than 10% had a vertical length of 42 cm (a reduction of 2 cm from Case 1)

and a horizontal length of 12 cm (an increase of 3 cm from Case 1). This distribution more closely resembled that of Case 1, which had no movement, than that of Case 2.

Although difference was observed in the distribution of inhaled air moving further from the torso of the CSP, the overall shape was similar that of Case 1. In Cases 4 and 5, the exhaled air, discharged at an approximate 45° downward angle, akin to other cases, and then exhibited a rising pattern due to the thermal plume's influence. Essentially, the analysis results indicated that linear movement of the CSP in the ML direction expanded the BZ in that direction, whereas movement in the AP direction resulted in a somewhat limited expansion in that aspect.

The comparison of maximum horizontal and vertical length in the BZ distribution provides a qualitative understanding of how micro-movement affects the BZ. However, to quantitatively compare the distinct differences in the inhaled air distribution, the volume of SVE 5 for each case is presented in Table 6.4, along with the percentage increase or decrease relative to Case 1. Overall, the volume of SVE 5 > 90% distribution concentrated around the nose decreased in all cases compared to Case 1, while the volume of SVE 5 > 10% increased. Specifically, in Cases 2 and 3, which simulate postural sway, the SVE 5 > 90% distribution decreased by more than 95% compared to Case 1, and the SVE 5 > 10% distribution increased by 7.84% and 9.89%, respectively. Although the overall range of SVE 5 expanded, the near disappearance of the SVE 5 > 90% distribution around the nose suggests that even if contaminants are present near the nose, the actual possibility of inhalation may decrease. This effect is particularly significant for particulate contaminants that are less influenced by concentration diffusion, emphasizing the importance of these changes in air distribution.

Table 6.4. Comparison of the SVE 5 volume according to the Case

	SVE 5 > 90% (Decrease compared to Case 1)	SVE 5>10% / (Increase compared to Case 1)
Case 1 (Stationary)	1.10.E-04 m ³ (-)	5.10e-04 m ³ (-)
Case 2 (Angular movement, 4.0°/s)	5.12.E-06 m ³ (95.34%)	5.51e-04 m ³ (7.84%)
Case 3 (Angular movement, 5.0°/s)	4.95.E-06 m ³ (95.50%)	5.62e-04 m ³ (9.89%)
Case 4 (Linear movement, ML)	3.49.E-05 m ³ (68.29%)	5.20e-04 m ³ (2.01%)
Case 5 (Linear movement, AP)	9.06.E-05 m ³ (17.67%)	5.79e-04 m ³ (13.73%)

The findings of this study highlight the critical role of replicating micro-movements in evaluating the quality of breathing air using a CSP. Accurately modeling the interaction between humans and their environments is essential for optimizing ventilation systems in confined spaces, such as Personal Work Environments (PWE) or personalized ventilation setups. However, reproducing the intricate shape and physiological functions of a CSP significantly extends the time and computational resources required for CFD analysis, necessitating a careful balance between cost-effectiveness and accuracy. The minimal influence of CSP's micro-movements on indoor airflow and thermal conditions indicates that for studies not specifically focusing on the detailed effects of human's micro-movement on indoor climates, omitting micro-movements could streamline computational efforts without substantially affecting the validity and accuracy of the research.

6.6. Limitations

This study has several limitations. Firstly, we employed a naked state CSP instead of applying the clothing model developed in previous chapters, to reduce the complexity associated with simulating breathing and movements. However, the airflow patterns created by the thermal plume

around the human body significantly impact the breathing zone, highlighting the need to consider clothing. Clothing can alter airflow patterns due to its uneven surfaces and changes in the surrounding thermal environment. Additionally, we applied displacement ventilation, which provides relatively stable airflow, but it remains essential to determine if micro-movements can significantly affect the breathing zone even in the presence of strong surrounding airflow. Furthermore, the range of posture movements varies with age and health, making it challenging to standardize different breathing rates and micro-movement patterns as done in this study. Future research will need to assess the impact of the breathing zone across various movements and evaluate the effects in a broader range of environments and movements.

6.7. Conclusion

In this chapter, we conducted a quantitative analysis of the impact of micro-movements on the BZ of a standing CSP under transient breathing conditions. Our comparisons of indoor airflow and temperature, with and without micro-movements, revealed that angular movements related to posture sway have minimally impact the overall indoor environment but significantly affect BZ formation. Notably, ML movements had a considerable effect on horizontal expansion of BZ. Thus, micro-movements directly influence BZ formation, essential for precise environmental assessments and improving ventilation in small spaces. However, in CFD-CSP studies on indoor environments and human exposure, maintaining analysis accuracy while considering cost-effectiveness is crucial. Applying movements to the CSP demands significant analysis time and resources, making it non-essential for studies focused mainly on the indoor environment but crucial for inhalation exposure assessments of contaminants where BZ is concerned.

REFERENCES

- [1] B.A. Edge, E.G. Paterson, G.S. Settles, Computational study of the wake and contaminant transport of a walking human, (2005).
- [2] Y. Tao, W. Yang, K. Inthavong, J. Tu, Indoor particle inhalability of a stationary and moving manikin, *Building and Environment* 169 (2020) 106545.
- [3] J. Hang, Y. Li, R. Jin, The influence of human walking on the flow and airborne transmission in a six-bed isolation room: tracer gas simulation, *Building and Environment* 77 (2014) 119–134.
- [4] S. Yoo, K. Ito, Validation, verification, and quality control of computational fluid dynamics analysis for indoor environments using a computer-simulated person with respiratory tract, *Japan Architectural Review*. 5 (2022) 714–727.
- [5] V.P. Panzer, S. Bandinelli, M. Hallett, Biomechanical assessment of quiet standing and changes associated with aging, *Archives of Physical Medicine and Rehabilitation* 76 (1995) 151–157.
- [6] C. Li, K. Ito, Numerical and experimental estimation of convective heat transfer coefficient of human body under strong forced convective flow, *Journal of Wind Engineering and Industrial Aerodynamics*. 126 (2014) 107–117.
- [7] F.R. Menter, M. Kuntz, R. Langtry, Ten years of industrial experience with the SST turbulence model, *Turbulence, Heat and Mass Transfer*. 4 (2003) 625–632.
- [8] D. Winter, Human balance and posture control during standing and walking, *Gait & Posture* 3 (1995) 193–214.
- [9] W.H. Gage, D.A. Winter, J.S. Frank, A.L. Adkin, Kinematic and kinetic validity of the inverted pendulum model in quiet standing, *Gait & Posture* 19 (2004) 124–132.
- [10] I.D. Loram, C.N. Maganaris, M. Lakie, Human postural sway results from frequent, ballistic bias impulses by soleus and gastrocnemius, *The Journal of Physiology* 564 (2005) 295–311.
- [11] K. Masani, A.H. Vette, N. Kawashima, M.R. Popovic, Neuromusculoskeletal Torque-Generation Process Has a Large Destabilizing Effect on the Control Mechanism of Quiet Standing, *Journal of Neurophysiology* 100 (2008) 1465–1475.
- [12] R.J. Peterka, Sensorimotor Integration in Human Postural Control, *Journal of Neurophysiology* 88 (2002) 1097–1118.

- [13] I.J. Pinter, R. Van Swigchem, A.J.K. Van Soest, L.A. Rozendaal, The Dynamics of Postural Sway Cannot Be Captured Using a One-Segment Inverted Pendulum Model: A PCA on Segment Rotations During Unperturbed Stance, *Journal of Neurophysiology* 100 (2008) 3197–3208.
- [14] L.A. Rozendaal, A.J. “Knoek” Van Soest, Stabilization of a multi-segment model of bipedal standing by local joint control overestimates the required ankle stiffness, *Gait & Posture* 28 (2008) 525–527.
- [15] I.D. Loram, C.N. Maganaris, M. Lakie, Human postural sway results from frequent, ballistic bias impulses by soleus and gastrocnemius, *The Journal of Physiology* 564 (2005) 295–311.
- [16] P. Morasso, A. Cherif, J. Zenzeri, Quiet standing: The Single Inverted Pendulum model is not so bad after all, *PLoS ONE* 14 (2019) e0213870.
- [17] S.S. Hasan, D.W. Robin, D.C. Szurkus, D.H. Ashmead, S.W. Peterson, R.G. Shiavi, Simultaneous measurement of body center of pressure and center of gravity during upright stance. Part II: Amplitude and frequency data, *Gait & Posture* 4 (1996) 11–20.
- [18] T.E. Sakanaka, M. Lakie, R.F. Reynolds, Idiosyncratic Characteristics of Postural Sway in Normal and Perturbed Standing, *Frontiers in Human Neuroscience* 15 (2021).
- [19] M.L. Latash, S.S. Ferreira, S.A. Wiczorek, M. Duarte, Movement sway: changes in postural sway during voluntary shifts of the center of pressure, *Experimental Brain Research* 150 (2003) 314–324.
- [20] H. Park, S.-J. Yoo, J. Seo, S. Eisaku, H. Hiroshi, K. Kuga, K. Ito, Integration of computer-simulated persons with multi-node thermoregulation model that considers the effect of clothing for skin surface temperature distribution analysis, *Building and Environment* 248 (2024) 111105.

CHAPTER 7

Summary and Future Work

7.1. Summary of the dissertation

This dissertation aimed to develop a comprehensive Computational Simulated Person (CSP) that can accurately predict and evaluate thermal comfort and human health risk in indoor environments, considering various scenarios at the design stage. First, a precise human-like CSP geometry and clothing model were established, and the CSP was integrated with the Fiala multi-node thermoregulation model to replicate human thermoregulation. Additionally, the thermal prediction accuracy of the CSP was validated by comparing the results with those from chamber experiments using the actual thermal manikin. Second, to precisely replicate human breathing function of the developed CSP, a nasal cavity model was integrated, and a transient breathing profile was applied. Third, continuous micro-movements that occur in stationary standing humans to maintain posture were replicated in this CSP. Using this, the impact of human micro-movements on the environment surrounding the CSP and the Breathing Zone was assessed and reported through coupled CSP-CFD analysis.

Chapter 1: Introduction

We clarified the background and purpose of the research and emphasized its necessity.

Chapter 2: Outline of numerical analysis

We explained the fundamental fluid equations necessary for calculating the flow field within the model targeted by this research, as well as the transport equations for scalar quantities like

thermal energy, moisture, and pollutants. It also described the turbulence models used and the radiative models for coupled heat transfer analysis, assuming moisture as a passive scalar for analysis. Using these supporting equations, the chapter detailed the numerical analysis performed on the non-uniform flow fields, temperature fields, humidity fields, and tracer gas concentration fields formed indoors, precisely assessing the impacts on the human body.

Chapter 3: CSP integrated with a multi-node thermoregulation model

We reported the creation process of the CSP's shape and clothing model based on the ANDI thermal manikin, which is widely used in experimental studies targeting indoor environments and the surrounding human body. Additionally, we introduced the Fiala multi-node thermoregulation model that controls the thermal functions and skin temperature of the thermal manikin in response to the surrounding environment. We integrated this model with the CSP-CFD coupled analysis.

Chapter 4: Validation of CSP's thermal prediction accuracy

We focused on validating the thermal prediction accuracy of the developed CSP integrated with multi-node thermoregulation model. The environment of the chamber experiment using ANDI thermal manikin was replicated precisely through CFD simulation, and then we conducted comparison of skin temperatures between the thermal manikin and CSP. The CSP analysis results reproduced the experimental results with acceptable accuracy, and the RMSD of the predicted average skin temperature was less than 0.3 in all cases. Furthermore, the importance of detailed clothing geometry was highlighted by confirming that the thickness of the ventilation layer between the skin and clothing has a significant effect on the thermal performance of the clothing.

However, in the cases with the seated CSP, the Pearson correlation coefficient did not show sufficient consistency, which necessitates further analysis to enhance reliability.

Chapter 5: Validation of CSP's thermal prediction accuracy

A nasal cavity model, derived from CT data of a healthy adult male's respiratory system, was integrated into the developed CSP, replicating the vital human function of breathing. This chapter introduces a transient breathing cycle model, simulating variable breathing rates over time to facilitate dynamic numerical analysis. Additionally, this chapter introduced SVE 4 and 5 used to identify the BZ distribution, which are crucial for evaluating contaminant transfer and assessing human inhalation exposure to potential contaminants.

Chapter 6: Micro-movement

We discussed the necessity of replicating micro-movements resulting from postural sway. To replicate these movements, we applied the single-mass Inverted Pendulum Model (IPM) to the CSP. Additionally, the chapter presents the results of CFD analysis that explored how micro-movements affect the formation of the CSP's BZ and affect the general indoor climate. When angular movement (single-mass IPM) was applied, the distribution of exhaled air remained generally consistent; however, distinct differences were observed in the distribution of inhaled air compared to the without movement case. Specifically, in a case with a maximum angular velocity of $4^\circ/\text{s}$, the horizontal length of inhaled air distribution expanded by 12 cm, and the vertical extent of the SVE5>10% distribution contracted by 25 cm compared to the stationary case. Notably, linear movement in the mediolateral (ML) direction resulted in a horizontal expansion of the inhaled air distribution. Additionally, it was confirmed that the micro-movements of the CSP have minimal impact on the distribution of the thermal plume surrounding

CSP and on the convective heat transfer coefficient (CHTC) of the CSP skin surface. The findings of this study suggest that replicating human micro-movements has minimal impact on general indoor climate analysis but significantly improves the accuracy of predicting air quality in breathing zones or assessing inhalation exposure to contaminants. However, replicating these micro-movements in numerical analysis significantly increases both the time and computational resources needed for CFD analysis. Therefore, whether to include such movements should be determined based on the specific needs and goals of the research.

7.2. Future work

In Chapters 3 and 4, various characteristics of clothing in the CSP were omitted, particularly, the porosity of clothing materials and the contact areas between the clothing and the CSP, which could potentially lead to an overestimation of clothing performance. Therefore, this aspect requires further research. Additionally, the environments of the chamber experiment and CFD simulation conducted in this study were set based on typical conditions without extreme environments or change. As such, factors like shivering or additional sweating were not considered. With this, further verifications are planned in more varied scenarios and environmental conditions to improve the CSP's applicability.

Chapter 6 introduces further experimental analysis and interpretation of micro-movements. Exploration of the effects of these micro-movements under diverse environmental conditions can be done. Furthermore, the primary objective of developing this CSP is not only to precisely evaluate indoor environments but also to assess the risks of human exposure. Therefore, the CSP model must be capable of quantitatively assessing both the responses and impacts of

contaminants on the human body. So, integration of a physiologically based pharmacokinetic (PBPK) model can also be added in the model. This will enable a more accurate modeling and give us further analysis of the pathways and outcomes of human exposure to various contaminants that may occur in indoor settings. Ultimately, this CSP can serve as a comprehensive risk assessment tool, capable of evaluating not only human respiratory exposure to airborne contaminants but also exposures through skin contact by adding dermal exposure models.

PUBLISHED ARTICLES RELATED TO THIS CHAPTER

Conference articles

- [1] H.G. Park, S.-J. Yoo, E. Sumiyoshi, H. Harashima, K. Ito, Indoor Thermal Comfort Assessment Using CFD-CSP Hybrid Analysis, (2022).
- [2] H.G. Park, S.-J. Yoo, E. Sumiyoshi, H. Harashima, K. Ito, Investigation of the impact of the detailed clothing modelling on the human thermal comfort based on multi-node thermoregulatory analyses using computer-simulated person, Healthy Building 2023 Asia Pacific, (2023).
- [3] P. Hyun-Gyu, Y. Sung-Jun, S. Eisaku, H. Hiroshi, K. Ito, Numerical Thermal Comfort Analysis Using Combined Computer-Simulated Person with Clothing and Multi-Node Thermoregulation Model, E3S Web of Conf. 396 (2023) 01019.
- [4] H.G. Park, S.J. Yoo, K. Kuga, K. Ito, The effect of micro-movement of a computer-simulated person on the formation of breathing zones, Indoor Air 2024 Conference.

Journal articles

- [1] S.J. Yoo, S. Yamauchi, H. Park, K. Ito, Computational Fluid and Particle Dynamics Analyses for Prediction of Airborne Infection/Spread Risks in Highway Buses: A Parametric Study, Fluids 8 (2023) 253. <https://doi.org/10.3390/fluids8090253>.
- [2] H.G. Park, S.J. Yoo, J. Seo, S. Eisaku, H. Hiroshi, K. Kuga, K. Ito, Integration of computer-simulated persons with multi-node thermoregulation model that considers the effect of clothing for skin surface temperature distribution analysis, Building and Environment 248 (2024) 111105.
- [3] H.G. Park, S.J. Yoo, S. Eisaku, H. Hiroshi, K. Kuga, K. Ito, Impact of human micro-movements on breathing zone and thermal plume formation, Building and Environment (2024) submitted.