Comprehensive Optimization of Air Quality in High Speed Rail Cabins Using CFD

A thesis submitted in fulfilment of the requirements for the degree of Doctor of Philosophy

Lin Yang
MEng University of Melbourne
BEng China Jiliang University

School of Engineering
College of Science, Engineering and Health
RMIT University

January 2019
Declaration

I certify that except where due acknowledgement has been made, the work is that of the author alone; the work has not been submitted previously, in whole or in part, to qualify for any other academic award; the content of the thesis is the result of work which has been carried out since the official commencement date of the approved research program; any editorial work, paid or unpaid, carried out by a third party is acknowledged; and, ethics procedures and guidelines have been followed.

I acknowledge the support I have received for my research through the provision of an Australian Government Research Training Program Scholarship.

Lin Yang

1st January 2019
Acknowledgements

This PhD would have not been possible without the help of many people inside and outside the university.

Firstly, I would like to thank my principal supervisor Prof. Jiyuan Tu who has kindly given me the opportunity to do a PhD under his critical advice, encouragement and immense knowledge in every stage of this work. His guidance helped me in all the time of my PhD research.

I would particularly thank my secondary supervisor Dr Xiangdong Li for tremendous academic contributions and countless support over my PhD study. He was keeping me on track, standing by me and always being there whenever I need.

Furthermore, I would also like to thank Dr Adeline Montlaur and Dr Jose I. Rojas for the opportunity to work with their group at Universitat Politècnica de Catalunya (UPC).

Many thanks to the academical support and company from my dear colleagues and friends in RMIT: Dr Yang Yuan, Dr Yidan Shang, Dr Yihuan Yan, Mr Nan Li, Mr Jiawei Ma, Mr Shuang Yi, Mr Xiang Fang, Dr Yao Tao, Dr Morteza Yousefi, Mr Zhan Zhao, Mr Yu Wang, Mr Saeed Dehghani and Dr Machimontorn Pro. To anyone I have forgotten to thank, I am sorry. However, it is just not possible to thank everyone who deserves it individually because there are so many of you.

I wish to offer my deepest gratitude to my parents, to whom this thesis is dedicated. They have motivated me in my life and have been role models for me to develop myself as an individual. Finally, I would like to thank my fiancé, my love and best friend, Rui Li for her confidence in me and unconditional support during these four years while undertaking her PhD at the same time.
Publication during Candidature

Peer Reviewed Journal Publication:


**Conference Publication:**


# Contents

Abstract ........................................................................................................................................... 1

Chapter 1  Introduction ................................................................................................................... 4  
  1.1 Motivation ............................................................................................................................... 4  
  1.2 Objectives ............................................................................................................................... 8  
  1.3 Thesis Outline ......................................................................................................................... 8  
  1.4 Contributions ......................................................................................................................... 10

Chapter 2  Literature Review ......................................................................................................... 12  
  2.1 Indoor Air Quality .................................................................................................................. 12  
    2.1.1 Gaseous Contaminants ................................................................................................. 12  
    2.1.2 Particle Contaminants ............................................................................................... 15  
  2.2 HST Ventilation Standard and Thermal Comfort Index ......................................................... 17  
    2.2.1 Ventilation Standard of Train Cabin ............................................................................. 17  
    2.2.2 Thermal Comfort Index ............................................................................................... 20  
  2.3 Influential Factors of Air Quality in HST cabins .................................................................. 23  
    2.3.1 Interior Impact Factors ................................................................................................. 23  
      2.3.1.1 Diffuser Design .................................................................................................... 23  
      2.3.1.2 Passenger Activities – Coughing Process ............................................................. 25  
    2.3.2 Exterior Impact Factors ................................................................................................. 27  
      2.3.2.1 HST Operation Conditions ................................................................................. 27  
      2.3.2.2 Solar Radiation .................................................................................................... 29  
  2.4 Optimization algorithm coupled with CFD .......................................................................... 31

Chapter 3  Methodology .................................................................................................................. 33  
  3.1 Project Breakdown .................................................................................................................. 33  
  3.2 Governing Equations for Airflow ......................................................................................... 34  
    3.2.1 Continuity Equations ................................................................................................. 35  
    3.2.2 Momentum Equations ............................................................................................... 36  
    3.2.3 Energy Equation ......................................................................................................... 36  
    3.2.4 Turbulence Modelling ................................................................................................. 37  
    3.2.5 The RNG k-epsilon Turbulence Model ........................................................................ 39  
  3.3 Governing Equations for Contaminants ............................................................................... 40  
    3.3.1 The Drift-flux Model for Gaseous Contaminants ......................................................... 40  
    3.3.2 The Eulerian-Lagrangian Model for Particulate Contaminants .................................. 41
6.2.3 Results and Discussion ........................................................................................................... 109
6.2.3.1 Ventilation Performance ..................................................................................................... 110
6.2.3.2 Containments Distribution .................................................................................................. 112
6.2.3.3 Ride Comfort ....................................................................................................................... 114
6.2.4 Conclusions ........................................................................................................................... 115

Chapter 7 The Effects of Solar Radiation on Thermal Comfort in HSR Cabins ..................................... 117
7.1 Introduction .................................................................................................................................... 118
7.2 Numerical Simulation ................................................................................................................... 119
7.2.1 Model Description ................................................................................................................... 119
7.2.2 Governing Equations ............................................................................................................. 123
7.2.3 Solar Load and Boundary Wall Details .................................................................................... 125
7.2.4 Thermal Comfort Index Analysis ............................................................................................ 127
7.3 Results and Discussion ............................................................................................................... 130
7.3.1 Validation of Numerical Model .............................................................................................. 130
7.3.2 Comparison of Simulation Results with and without Solar-radiation .................................... 134
7.3.3 The Solar-Radiation Effects at Different Time in One Day .................................................... 138
7.3.4 The Solar-Radiation Effects with Applying the Roller Curtain ............................................. 141
7.4 Conclusions ............................................................................................................................... 143

Chapter 8 Multi-objective Optimization of Air Quality in HSR Cabins .............................................. 145
8.1 Introduction .................................................................................................................................... 146
8.2 Methods ...................................................................................................................................... 148
8.2.1 Particle Swarm Optimization (PSO) and Nondominated Sorting-based PSO ........................... 148
8.2.1.1 Nondominated Sorting ....................................................................................................... 150
8.2.1.2 Crowding Distance ........................................................................................................... 150
8.2.2 Kriging and Multi-fidelity Kriging .......................................................................................... 151
8.2.3 Workflow of the Design Procedure ......................................................................................... 154
8.3 Computational Model Description ............................................................................................. 155
8.3.1 Model Preparation and Validation .......................................................................................... 155
8.3.2 Design Objectives ................................................................................................................... 157
8.4 Results and Discussions ............................................................................................................. 159
8.4.1 Design Optimization Problem for the HST Cabin Ventilation ............................................. 159
8.4.2 Validation of the High-fidelity Kriging Method ........................................................................ 163
8.4.3 Design Optimization Using NSPSO with High-fidelity Kriging ........................................... 165
8.4.4 Enhancement in Optimization Process Using the Multi-Fidelity Kriging .............................. 167
8.4.5 Multi-fidelity Training Samples for HST cabin Ventilation Problem .................................... 169
8.5 Conclusions ............................................................................................................................... 173
Chapter 9   Conclusions .................................................................................................................. 175

9.1 Summary of the Contributions .......................................................................................... 176

9.1.1 The Effects of Diffuser Type on Thermal Flow and Contaminant Transport in HSR Cabins 176

9.1.2 The Effects of Cough-jet on Airflow and Contaminant Transport in a Cabin Section ..... 177

9.1.3 The Effects of Induced Pressure Fluctuations on Air Quality in HSR Cabins ............... 178

9.1.4 The Effects of Solar Radiation on Thermal Comfort in HSR Cabins ......................... 179

9.1.5 Multi-objective Optimization of Air Quality in HSR Cabins ..................................... 180

9.2 Summary of Simulation Models ......................................................................................... 180

Bibliography .............................................................................................................................. 182
List of Figures

Figure 1.1 Preferred HSR alignment and stations for the east coast of Australia (AECOM, 2013) ........................................................... 5

Figure 2.1 Relative size chart of common air contaminants (Howie, 1990) ................................. 15

Figure 2.2 Particle deposition in nasal cavity ............................................................. 16

Figure 2.3 The PPD index as the relation to PMV index ................................................. 22

Figure 2.4 Popular types of cabin diffuser design ......................................................... 25

Figure 2.5 Cough-jet experiment setup explanation (up-left); Cough-jet experiment rig (up-right); Cough-jet velocity profile (down-left); Cough-jet particle size discretion (down-right) .......................................................... 27

Figure 2.6 Exterior pressure waves when HST entering a tunnel (Yang, 2018) ....................... 29

Figure 2.7 Solar radiation on the HST cabin (up-left); Solar radiation through the HST cabin window (up-right); Thermal balance of HST cabin when considering solar radiation(down) .................................................................................................................. 30

Figure 3.1 Schematic view of breaking down the project into separate components ................. 34

Figure 4.1. A typical ventilation scheme in HST cabins .................................................. 48

Figure 4.2. Comparison of velocity field between experiment and simulation in front of one-row passengers (Li et al., 2015b) .................................................................................................................... 54

Figure 4.3. Comparison of velocity and temperature between experiment and simulation in 3 locations (Li et al., 2015b) .................................................................................................................... 55

Figure 4.4. Diffuser types and locations ........................................................................... 56

Figure 4.5. Simulation cabin and containments releasing source ...................................... 57

Figure 4.6. Velocity and Temperature fields ...................................................................... 59

Figure 4.7 Normalised VOCs concentration .................................................................... 62

Figure 4.8 VOCs concentration at 0.05 isosurface .......................................................... 63

Figure 4.9 Particle transport and distribution at various time steps ................................. 66

Figure 5.1 Simulation cabin section and the grid size around the mouth area ...................... 72

Figure 5.2 Cough-jet flowrate of a single cough based on experimental testing (Gupta et al., 2009) .................................................................................................................... 74
Figure 5.3 Comparison of airflow vectors between experimental data and computational results (Yan et al., 2015) ................................................................. 76
Figure 5.4 Velocity vector field around breathing-zone for cough passenger B in Case-2. .............. 78
Figure 5.5 Velocity vector field across X-plane for different cough passenger.................... 79
Figure 5.6 Particle trajectories for without cough-jet and with cough-jet cases. ..................... 81
Figure 5.7 Particle traveling distance in 13s for without cough-jet and with cough-jet cases ...... 82
Figure 5.8 Particle number and mass distributions of various sizes. ........................................ 83
Figure 5.9 Particle transport and distribution of various sizes................................. 84
Figure 5.10 Average particle residence time in the passengers’ breathing zones.................. 85
Figure 6.1 Typical pressure waves induced by a high-speed train traveling in a tunnel (Ricco et al., 2007) .................................................................................. 89
Figure 6.2 The cabin mock-up model for model validation (Yan et al., 2017) ......................... 92
Figure 6.3 Comparison of the predicted airflow vectors against the PIV image (Yan et al., 2017) ........................................................................................................ 92
Figure 6.4 Comparison of the air velocity profiles along selected vertical lines ................. 93
Figure 6.5 The computational domain: The overall view of the computational domain (Up); The CRH-2 cabin interior (Down left); Interior view of the computational domain (Down right).......................................................................................... 94
Figure 6.6 Instantaneous pressure fluctuations measured onboard a CRH-2 cabin (Liu, 2012) ... 95
Figure 6.7 The temperature and airflow fields in the cabin without pressure fluctuation.......... 97
Figure 6.8 Map of the predicted mean vote (PMV) for each passenger .............................. 98
Figure 6.9 Mean age of air (MAA) at Plane Y1................................................................... 99
Figure 6.10 Normalized contaminant concentration at Plane Y1 ........................................ 100
Figure 6.11 The air pressure measured near the passenger ears ........................................ 100
Figure 6.12 The air pressure contour in Plane Z1 at t = 2.0 s .............................................. 101
Figure 6.13 Effects pressure fluctuation on the thermal flow field in the cabin: Air velocity (Up); Air temperature (Down) ........................................................................ 102
Figure 6.14 The effects of pressure fluctuation on the contaminant concentration field were negligibly small................................................................................... 102
Figure 6.15 The influence of pressure fluctuation on the PMV of passengers ...................... 103
Figure 6.16. Simulation cabin and ventilation boundaries................................................... 106
Figure 6.17. Induced pressure fluctuations measured when a HST passing through a 1km tunnel (Liu, Chen 2017) ................................................................. 107

Figure 6.18 Pressure gradient from experimental and simulation results ........................................ 109

Figure 6.19 Mean temperature (Up) and velocity (Down) profiles at a selected plane in different time-steps .................................................................................................. 111

Figure 6.20 Temperature (Up) and velocity (Down) above manikins seated in the third row at different time steps ........................................................................................................ 112

Figure 6.21 VOCs concentration rendering around released passenger in steady-state .............. 113

Figure 6.22 The changing range of PMV value for each passenger during the 18 seconds .... 114

Figure 6.23 Average hearing pressure during the 18 seconds ......................................................... 115

Figure 7.1 A typical HST cabin model .......................................................................................... 120

Figure 7.2 Mesh independence test ............................................................................................. 121

Figure 7.3 Surface cluster sensitivity test .................................................................................... 124

Figure 7.4 Solar radiation characters (a. change of solar radiation intensity during the day; b. direction of solar radiation) .................................................................................... 126

Figure 7.5 A typical roller curtain used in HST cabin ................................................................. 127

Figure 7.6 Comparison with Rodler's experiment (a. solar intense on the floor from experiment; b. solar intense on the floor from simulation; c. temperature contour of the floor from experiment; d. temperature contour of the floor from simulation) .......................... 131

Figure 7.7 Comparison with Gendelis’ simulation (a. comparison of temperature contour on the floor; b. comparison of velocity vector in the cross-section plane; c. comparison of the velocity magnitude on the red line) ........................................................................ 133

Figure 7.8. The comparison of temperature and velocity in the Plane Z1 (a. without solar-radiation; b. with solar-radiation at 14:00; c. The temperature profile on one typical line) ................................................................................................. 134

Figure 7.9. The solar heat flux on the surfaces (a. The absorbed solar heat flux by cabin interior walls; b. The absorbed solar heat flux by manikins and seats) ................................................. 136

Figure 7.10. The distribution of PMV index (a. Without solar-radiation; b. With solar-radiation at 14:00) ................................................................................................................. 137

Figure 7.11. The distribution of PPD index (a. Without solar-radiation; b. With solar-radiation at 14:00) ................................................................................................................. 138

Figure 7.12. Surface temperature of two passengers seating next to the window ...................... 139

Figure 7.13 Thermal comfort index change with time profiles (a. Average manikin surface temperature; b. Average cabin air temperature; c. Average PMV value; d. Average PPD value; e. Average PD value.) ............................................................... 141
Figure 7.14. Comparisons of temperate and PMV contour for passengers exposed to the solar ray

Figure 8.1 An example of the nondominated sorting process in NSPSO.

Figure 8.2 Crowding distances among individuals in the highest-level nondominated front.

Figure 8.3 Framework of the CFD-based multi-objective optimization approach with Multi fidelity Kriging.

Figure 8.4 Cabin geometry and detailed meshing features.

Figure 8.5 Comparison of temperature and velocity between experimental and simulation results.

Figure 8.6 Definitions of inlet boundary conditions in CFD simulations.

Figure 8.7 CFD simulation results of case 1 in Table 8.1.

Figure 8.8 CFD simulation results of case 2 in Table 8.1.

Figure 8.9 CFD simulation results of case 3 in Table 8.1.

Figure 8.10 The additional CFD simulations and its corresponding ventilation conditions within the design space (i.e. green points).

Figure 8.11 Percentage errors of Kriging prediction for the three objectives - PMV, Contaminant and Energy, respectively.

Figure 8.12 Optimal solutions (‘Pareto Front’) given by NSPSO in objective space.

Figure 8.13 Probability density of optimal solutions in design space.

Figure 8.14 An example of 1-D Multi-fidelity Kriging prediction results. (a) The prediction using only a few of high-fidelity data is undersampled and there are huge differences between the prediction values and the true values. (b) The prediction using a large amount of high-fidelity data is well sampled. (c) The prediction using a few of high-fidelity data plus a large amount of low-fidelity data achieves good prediction accuracy, where the low-fidelity data are inaccurate but help to correct the prediction.

Figure 8.15 Mesh independent testing results.

Figure 8.16 The arrangement of low and high-fidelity training samples for Multi-fidelity Kriging.

Figure 8.17 Comparison of prediction differences between Multi-fidelity Kriging and fully high-fidelity Kriging.

Figure 8.18 Comparison of optimal solutions (‘Pareto Front’) in objective space between using original high-fidelity Kriging and Multi-fidelity Kriging.
List of Tables

Table 2.1 VOCs compounds in different places of human body (Lundström and Olsson, 2010) .................................14
Table 2.2 Outside environment parameters for calculation ..............................................................18
Table 2.3 Cabin interior temperature standards for the passenger seating area ..........................18
Table 2.4 Cabin interior airflow velocity standard for the passenger seating area ......................19
Table 2.5 Cabin fresh air flow rate standard for the passenger seating area ..............................19
Table 2.6 PMV thermal sensation scale ............................................................................................21
Table 4.1. Summary of HST cabin HVAC operation requirements in summer (MRPRC, 1987, UIC, 2005) .........................................................50
Table 4.2. PMV value of four type diffusers used cabin ........................................................................60
Table 4.3. VOCs concentration in breathing zone of four type diffusers used cabin ..................63
Table 4.4. Particles averaged volume fraction in breathing zone of four type diffusers used cabin ........................................................................67
Table 5.1 Discrete probability of particle size during one coughing ..............................................74
Table 6.1 VOC relative concentration ratio in high concentration positions: P8B (seat next to source manikin) and P7C (seat in front of source manikin) ................................................113
Table 7.1 Material Properties of the HST cabin and passenger .....................................................121
Table 7.2 Comparison of thermal comfort index for passenger exposed to the solar ray ..............143
Table 8.1 Three typical groups of values minimizing PMV, Contaminant, Energy, respectively161
Table 8.2 Comparisons of CPU time consumptions (100 particles in the optimization process) 173
Table 9.1 Summary of simulation models ......................................................................................181
### Nomenclature

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>( \alpha )</td>
<td>Thermal diffusivity</td>
</tr>
<tr>
<td>( \Gamma )</td>
<td>General diffusion coefficient</td>
</tr>
<tr>
<td>( \lambda )</td>
<td>Thermal conductivity</td>
</tr>
<tr>
<td>( \mu )</td>
<td>Dynamic Viscosity</td>
</tr>
<tr>
<td>( \mu_{\text{eff}} )</td>
<td>Effective viscosity</td>
</tr>
<tr>
<td>( \mu_t )</td>
<td>Turbulent viscosity</td>
</tr>
<tr>
<td>( \varepsilon )</td>
<td>Dissipation rate of turbulent kinetic energy</td>
</tr>
<tr>
<td>( \varepsilon_k )</td>
<td>Emissivity of surface ( k )</td>
</tr>
<tr>
<td>( \rho )</td>
<td>Density</td>
</tr>
<tr>
<td>( \sigma )</td>
<td>Stefan-Boltzmann constant</td>
</tr>
<tr>
<td>( \phi )</td>
<td>Scalar variable for governing equation</td>
</tr>
<tr>
<td>( \phi' )</td>
<td>Fluctuating component of scalar variable</td>
</tr>
<tr>
<td>( \varphi )</td>
<td>Concentration of ( g ) gaseous contaminant</td>
</tr>
<tr>
<td>( A_k )</td>
<td>Area of surface ( k )</td>
</tr>
<tr>
<td>( C_D )</td>
<td>Drag coefficient</td>
</tr>
<tr>
<td>( C_p )</td>
<td>Thermal capacity</td>
</tr>
<tr>
<td>( D_\varphi )</td>
<td>Kinematic diffusivity of gaseous contaminant</td>
</tr>
<tr>
<td>( d_p )</td>
<td>Particle diameter</td>
</tr>
<tr>
<td>( F_B )</td>
<td>Buoyancy force</td>
</tr>
<tr>
<td>( F_D )</td>
<td>Drag force</td>
</tr>
<tr>
<td>( F_{kj} )</td>
<td>View-factor between surface ( k ) and ( j )</td>
</tr>
<tr>
<td>( f_{cl} )</td>
<td>Clothing surface area factor</td>
</tr>
</tbody>
</table>
\( g \)  
Gravity vector

\( h \)  
Convective heat transfer coefficient

\( I \)  
Intensity of the turbulence flow

\( k \)  
Turbulent kinetic energy

\( m_p \)  
Mass of particle

\( PD \)  
prediction of percentage of dissatisfied people due to draft

\( PMV \)  
Predicted Mean Vote

\( PPD \)  
prediction of percentage of dissatisfaction of the thermal comfort

\( Pr \)  
Prandtl numbers

\( p \)  
Pressure

\( Q \)  
Air flow rate

\( q_k \)  
Energy flux of surface k

\( Re \)  
Reynolds number

\( R_{ij} \)  
Reynolds stress tensor

\( S_\theta \)  
Source term

\( S_T \)  
Internal thermal source

\( T \)  
Temperature

\( t \)  
Time

\( U_p \)  
Particle velocity

\( u, v, w \)  
Velocity in x, y, z component

\( V \)  
Velocity
Abstract

In recent years, High-speed rails (HSR) have been rapidly developed in many counties due to convenience and high efficiency. A growing number of people choose High-speed train (HST) for long-distance travel. The good air quality in train cabin relates to the HVAC performance, which can optimize passengers’ comfort zones, efficiently control the gas/particle contaminants transportation inside a cabin. Thus, in-cabin air quality is of great interest to many researchers. The air quality is affected by many factors such as ventilation scheme, cabin geometry, contaminant property, train operation condition, and ambient environment. The focus of this project is to comprehensively and effectively assess the air quality in HST cabins and propose corresponding optimization strategies.

Computational Fluid Dynamics (CFD) has been proven as a cost-efficient approach to analyze and optimize air quality of indoor environments. However, the holistic optimization of air quality in HST cabin is still absent in existing literature due to the extreme complexity of the impact factors, which are dependent on unknown relationships. The HST cabin environment is influenced by many factors, ranging from cabin interior to exterior. Considering the interior, diffuser design is key to the ventilation performance, which can induce the overall ventilation flow pattern in a train cabin. Additionally, passengers’ activities from the interior cabin can also influence the air quality, such as coughing droplets from passenger which are the main source of contaminants transportation. On the other hand, train operation conditions and ambient environment are the main exterior impact factors for air quality. High-pressure waves are induced in two situations: when crossing another HST and when passing through a tunnel. Though these two situations always happen during HST operation, the study on the impact of the induced pressure waves on HST interior air quality is rare. Besides, considering HSTs are significantly exposed to solar radiation, the change of interior air quality due to solar radiation cannot be neglected. In order to optimize the air quality in HST, it is necessary to consider various objectives such as thermal comfort,
containment control, and energy consumption. Therefore, an integrated optimization approach is expected.

The main body of this thesis was composed of nine chapters. In the first two chapters, research background was introduced, and literature were summarized with research gaps identified. Research methodology was explained in Chapter 3. Then, the main research contributions were demonstrated from Chapters 4 to 8. Specifically, Chapter 4 and 5 discussed two impact factors with regard to the HST interior. In Chapter 4, four types of popularly-used diffusers were studied and compared regarding their ventilation performance and contaminate control. Chapter 5 investigated the transient coughing process released from passengers. This was the main source of particle contaminants in a densely occupied train cabin. For the HST exterior, the induced pressure fluctuation and solar radiation effects were presented in Chapter 6 and Chapter 7 respectively. The change of interior air quality when two HSTs passing by each other and when the HST passing through a tunnel were analyzed in Chapter 6. Chapter 7 demonstrated the influence of solar radiation on the HST cabin environment. Simulation under different setups including various daytime durations and with curtain applied situations were studied. As all the above-mentioned interior and exterior impact factors were found having a strong influence on the air quality in HST cabin, a multi-objective optimization algorithm was introduced in Chapter 8 to find the suitable ventilation solution based on user requirement. All the findings presented in chapters 4 to 8 were concluded and highlighted in Chapter 9, followed by a list of all the published work during the PhD candidature period.

In summary, this thesis presents an investigation of the HST interior air quality. This research contributed to the following outcomes: (a) A comprehensive understanding of different type of diffuser effect on HST cabin ventilation performance, thermal comfort and containment dispersion processes; (b) A preciously cough-jet model to represent the transport and distribution of cough-generated airborne contaminants under the cabin environment; (c) A systematic CFD model of the interior air response to the induced pressure waves during the period of two HSTs passing by each other and the HST passing through a tunnel; (d) A quantizable approach to assess the solar radiation effect
on thermal comfort in a HST cabin; (e) An efficient approach to manage the multi-objective optimization in HST cabin ventilation system. The computational studies presented in this thesis lay a solid foundation for air quality optimization and health risk assessments in HST cabin environment, which can be also applied in other densely occupied spaces such as metro, bus, and airline. Meanwhile, the outcomes of this study can be a supplement to the current industry standards.
Chapter 1

Introduction

1.1 Motivation

High-speed trains (HSTs) play an important role in domestic and international passenger transportation thanks to their unparalleled transport capacity, high efficiency, and strong reliability of being operated in most weather conditions. Inspired by the success of French TGV, Japanese Shinkansen and recent Chinese Railway High-speed Train (CHR), an increasing number of countries including Australia have been planning to develop their own high-speed rail (HSR) systems, as see in Figure 1.1 (AECOM, 2013). It is predicted that future HSTs will be the major competitor against airliners in the sector of long-distance passenger transportation (Albalate et al., 2015).

Like other public transportation vehicles, when HST travel is bringing convenience and efficiency to societies and boosting the railway industry, great concerns on the cabin air quality are also raised. The HST cabins are usually very crowded, especially in the economy class where passengers sit close to each other and moving space is limited. Also, the HST cabin environment is relatively enclosed (Chow, 2002). Passengers travelling in such a highly-occupied and enclosed environment over a long period of time would inevitably feel uncomfortable or even sick, whereas they are unable to
leave during the trip. Therefore, increasing attention has been drawn on the air quality in HST cabins, especially on the thermal comfort and disease transmissions.

Figure 1.1 Preferred HSR alignment and stations for the east coast of Australia (AECOM, 2013)

Existing studies have found a wide variety of contaminants which can be released or generated from multiple sources in the cabin environment. The contaminants suspending in the train cabin include not only gaseous contaminants, such as Volatile Organic Compounds (VOCs), Oxidation products, but also particulate contaminants, such as airborne droplets, virus, and bacteria (Mangili and Gendreau, 2005, Coleman et al., 2008). Among these contaminants, pathogen-carrying saliva droplets and nuclei released through cough or sneeze by an infected passenger have been emphasised in many epidemiology reports (Kenyon et al., 1996, Mangili and Gendreau, 2005). During past years, there have been a large amount of travel-related global outbreaks such as Tuberculosis (TB), Severe Acute Respiratory Syndrome (SARS) and Swine Influenza (H1N1) (Zhu et al., 2010). Since HST passengers sit densely in an enclosed and limited space, these contaminants are more possible to be inhaled by other occupants compared with other indoor environments such as offices and dwelling houses. As the perniciousness of the saliva droplets has been widely raised, the understanding of their
transport characteristics in HST cabin environment is necessarily required to precisely predict the infection risks to the individual passenger.

Computational Fluid Dynamics (CFD) is one of widely used methods to investigate airflow and evaluate contaminant transport and distribution in HST cabin environments, because it has the advantages such as time and cost reduction, the possibility of generating different graphs, and clear visualisations which facilitate the understanding of result features. CFD is also able to provide full-scale train cabin simulations, visualise the contaminants transport processes, and lead to an in-depth understanding of the complicated physical phenomena (Nielsen, 2015). For contaminant transport in indoor spaces, two distinct approaches namely the Eulerian model and the Lagrangian model have been employed in CFD simulations. So far, many studies have proved the capability of CFD study in evaluating the cabin environment (Stancato et al., 2015) and the exposure to contaminants (Poussou et al., 2010).

According to Han (Han and Huang, 2005), the in-cabin environment is affected by a large number of parameters such as interior air temperatures, air velocity and its profiles over different geometries, relative humidity, solar intensity and its incidence angles, different type of ventilation equipment, change of exterior environment, etc. More importantly, most of these parameters are dependent with uncertain relationships. In order to comprehensively understand the influence on HST cabin air quality and thermal comfort, these impact factors have been classified as interior cabin impact factors and exterior cabin impact factors.

Inside a HST cabin, the cabin design can directly influence the cabin air flow. One of the most important parts of cabin design is the diffuser system. According to (Chow and Wong, 1994), the diffuser locations can influence passengers’ thermal comfort, and improper diffuser orientation angles may result in higher temperatures in some areas. In today’s market, various types of cabin diffuser are available such as linear diffusers, round diffusers, grid diffusers and personal diffusers. The efforts of each type of diffuser have not been fully investigated and compared. Another important factor is passengers’ activities such as the body odour from a passenger or the cough-jet from a passenger’s mouth. Studies found an adult in rest state would emit an average of
14.8mg VOCs (body odor) per hour (Batterman and Peng, 2010). In addition, particle contaminants, infectious virus for instance, are released through coughing. Even though the cough-jet only lasts for a few seconds, it was claimed as an important factor for the local airflow distribution and the particle contaminates transportation (Gupta et al., 2009). Thus, to optimize the air quality in HST cabin, the interior impact factors should first to be investigated.

Besides, the HST operation conditions can also influence the air quality in the cabin. Since HSTs are operated at a speed between 200 km/h to 500 km/h, strong disturbance exists in surrounding air outside train cabins (Zhao et al., 2013). Especially when two HSTs are passing by each other or the HST passing through a tunnel, the disturbance is aggravated which may induce large transient pressure fluctuations. According to (Liu and Li, 2012, Zhang and Li, 2012), the induced pressure waves could also influence the cabin internal air environment through the ventilation system and even through door gaps. Another important exterior influential factor is solar radiation. HSTs are usually operated in open air where they are directly exposed to the solar load. The process that radiative rays transmit through windows into HST cabins is very complex. It relates to solar emitting angle, radiation intense, local time, location coordinates, weather condition, train travelling direction et al. Therefore, considering the complexity of HST operation conditions, it is necessary to assess the air quality in HST cabins under individual exterior influential factors.

As aforementioned, not only interior but also exterior factors that affect the air quality in HST cabins. Thus, the optimization of HST cabin environment is a complex process. Multiple evaluation criteria are involved such as air velocity, temperature near occupants, contaminant concentrations, percentage dissatisfied of draft, age of air, total energy consumption, etc. The system design is therefore a multi-objective optimization process, while the trade-off relations among those design indices are usually needed to be considered. For example, in hot summer, by reducing the temperature and increasing the velocity of air at inlet diffuser, the passengers’ thermal comfort can be improved but more energy needs to be consumed. Therefore, if a multi-objective optimization algorithm which can be conjunct with the CFD simulations was proposed, it can
provide a flexible design, which supports the selection of alternative solutions with the emphasis of a trade-off among the conflicting parameters.

With the research gaps identified, this thesis aims to provide an overall understanding of the influential factors on HST cabin air quality. Also, based on the investigated influential factors, a multi-objective optimization algorithm is proposed.

1.2 Objectives

The primary goal of this study is to develop the numerical model which can comprehensively optimise and assessment the air quality in HSR cabins. In order to achieve this goal, the following sequential activities have been conducted:

- Obtain fundamental knowledge of ventilation scheme in relation to mechanistic and physical environments.
- Develop an integrated CFD model which is capable of giving a full description and an accurate prediction of the airflow and contaminants transport in HSR cabins.
- Analyze the interior and exterior impact factors of HST cabin air quality.
- Optimize the cabin air quality with multi-objectives to fulfill different user requirements.
- Give design advice of optimized ventilation scheme to the public transport industry.

1.3 Thesis Outline

This thesis is composed of nine chapters. The aim of Chapter 1 is to provide a brief description of whole work, started with the background and motivation of the research in air quality in HST cabin. The objectives of this research work are explained subsequently. The focuses for following chapters are outlined below:

Chapter 2 provides a comprehensive literature review in relation to the existing researches of air quality in the HST cabin environment. The main airborne contaminants in train cabins are carefully reviewed and the thermal comfort index used in industry are summarised. The main affecting factors of air quality in HST cabins, including diffuser design, cough-jet, induced pressure wave, and solar radiation are
presented. The multi-object optimization algorithm used for HST air quality is also reviewed and discussed. The reviewed literature lays a solid foundation for the research outcomes in the following chapters.

Chapter 3 illustrates the detailed strategy of the project and the applied methodologies in the following chapters. The procedure of this thesis has been outlined, which including the explanation of breaking down the whole project into multiple components to individually investigate the major components and obtain the fundamental knowledge of each component. Meanwhile, the mathematical models of airflow and contaminants transport were explained in detail. Eventually, the radiation model and thermal comfort index used for the densely occupied HST cabin were presented.

Chapter 4 evaluates the importance of cabin diffuser effect in HST cabin environment, which is the key point to the cabin ventilation system. CFD models containing four different diffuser types were developed. The drift-flux model and Lagrangian model were employed to predict gaseous and particulate contaminants transport trajectories. The outcomes proved that the diffuser design plays a pivotal role in cabin ventilation system, thermal comfort, and contaminants transport.

Chapter 5 investigates the effect of cough-jet on local airflow and contaminant transport in a typical cabin environment by using CFD. Contaminants were released through coughing passengers from different locations. The transient numerical results in terms of contaminant transport characteristics were examined and compared. The comparison results revealed that cough-jet has significant effects on air flow in a certain range and time. Thus, this approach is highly recommended for future utilisation in the densely occupied environment.

Chapter 6 tests the response of cabin interior airflow, contaminant transport and thermal comforts to the transient pressure fluctuation during HSTs passing each other and passing through a tunnel. The HST cabin used in this study operates at a speed of 300 km/s, with the exterior pressure wave intrude into the cabin through the door-daps or the air-conditioning vents. For the situation that two HSTs passing each other, the conclusion shows transient pressure fluctuation only affects passengers’ hearing
pressure in a very short time. But for the situation that HST passing through a tunnel, the transient pressure fluctuation could influence the ventilation performance, air quality and thermal comfort in the area near doors.

Chapter 7 focuses on the mechanistic study of solar radiation effects on thermal comfort in a typical HST cabin. Parametric studies with 7 different daytime hours were carried out. The effects of using the roller curtain were also studied. It was found that overall cabin air temperature, especially near passengers, significantly increases with the presence of solar radiation. To improve the ride comfort and reduce energy consumption in hot weather, using a curtain could effectively reduce the solar radiation effects in the cabin environment.

Chapter 8 demonstrates a multi-objective optimization platform using nondominated sorting-based particle swarm optimization (NSPSO) algorithm to perform multi-objective optimal design of the ventilation system in a fully occupied HST cabin. Different combinations of ventilation operation parameters were investigated against its performance in terms of cabin thermal comfort, air quality and energy consumption. A Multi-Fidelity Kriging surrogate procedure is also proposed to reduce computational cost by replacing part of fine mesh CFD simulations with coarse mesh ones while maintaining acceptable accuracy. It demonstrates that in HST ventilation system design, the presented approach can deal with multi-objective optimization more efficient with up to 35.61% reducing of simulation time.

Chapter 9 presents the conclusion of this thesis by summarizing the outcomes from chapter 4 to chapter 8 and discusses further investigations required.

1.4 Contributions

In addition to the knowledge provided by the previous researchers, this thesis contributes to the following major outcomes:

- A comprehensive understanding of different type of diffuser effect on HST cabin ventilation performance, thermal comfort and containment dispersion processes.
• A precisely cough-jet model to represent the transport and distribution of cough-generated airborne contaminants under the cabin environment.

• A systematic CFD model of the interior air response to the induced pressure waves during the period of two HSTs passing by each other and the HST passing through a tunnel.

• A quantizable approach to assess the solar radiation effect on thermal comfort in a HST cabin.

• An efficient approach to managing the multi-objective optimization in HST cabin ventilation system.

The computational studies presented in this thesis lay a solid foundation for air quality optimization and health risk assessments in HST cabin environment, which can be also applied in other densely occupied spaces such as metro, bus and airline. Meanwhile, the outcomes of this study can be a supplement to the current industry standards.
Chapter 2

Literature Review

2.1 Indoor Air Quality

During the travel, passengers are long time stay inside the HSR cabin. To maintain the good air quality inside the cabin is a critical issue. Air contaminants inside the relative closed cabin environment would downgrade indoor air quality (IAQ) and affect passengers’ comfort and even health (Wolkoff, 1991, Franklin, 2007). According to the existing studies (Kelly and Fussell, 2015, Austin et al., 2002), the indoor air quality can be affected by different types of contaminants, such as pollen, biological particles, smoke and virus. As classified by previous researcher (Austin et al., 2002), the current found indoor contaminants can be mainly summarised into two categories: gaseous contaminants and particulate contaminants (which includes biological contaminants). Among currently found indoor contaminants, some contaminants such as the virus can directly affect passengers’ health, while other contaminants like body odour are nontoxic but could cause unpleasant sensory (Wolkoff, 2013). In this section, the airborne contaminants were discussed from both gaseous and particle contaminants.

2.1.1 Gaseous Contaminants

Gaseous contaminants are the most common type of cabin airborne pollution, which include hundreds of VOCs and inorganic gaseous contaminants. Due to the extreme
small size, gaseous contaminants have distinctive properties when compared with the other category of indoor air contaminants. They are existing in the form of individual molecules like nitrogen and oxygen in the indoor environment. The size for these molecules is normally between $3.4 - 20 \times 10^{-10}$ meters. The molecules of gaseous contaminants have fast thermal motion speed in random directions. The dynamic movement of indoor gaseous contaminants is governed by convection and diffusion laws.

Considering the component of gaseous contaminants, two sub-categories can be further classified: VOCs and inorganic gaseous contaminants. An inorganic compound is typically a chemical compound that lacks C-H bonds. But for organic compounds, it contains least one carbon atom, such as oxygen, hydrogen and nitrogen. However, in a normal indoor environment such as HSR cabin environment, the VOCs are the predominant gaseous contaminants. This is because the inorganic gaseous contaminants such as CO and H$_2$S are rare to be found in HSR cabin. Thus, the gaseous contaminants in this study would mainly focus on the VOCs.

From the previous study (Mirkhani et al., 2012), more than five hundred kinds of gaseous contaminants can be detected in the indoor environment. However, the total concentration of these gaseous contaminants in the indoor environment is very small. In a normal indoor environment, the concentration of VOCs is less than 1 milligram per cubic metre. Even the concentration of VOCs in the indoor environment is very small, there are many sources can release VOCs, such as the material (carpet, vinyl floors, composite wood and upholstery fabrics), appliances and personal care products (cooked food, fuel oil, Air fresheners et al.) and people. Among these VOCs releasing sources, though the new materials may release VOCs, their emission rate decline quickly over time. For old materials, the emission rate of VOCs is negligible. The appliances and personal care products only release VOCs when in use. But for passengers, the VOCs release rate is continuous during all the travel. Therefore, in the HST cabin environment, the VOCs are mainly released from passengers.
According to the study (Lundström and Olsson, 2010), the VOCs formatted from human beings are mainly caused by bacterial activity and skin glands excretions. Table 2.1 present the type of VOCs compounds in different places of human body.

Table 2.1 VOCs compounds in different places of human body (Lundström and Olsson, 2010)

<table>
<thead>
<tr>
<th>Part of body</th>
<th>Examples of Compounds</th>
</tr>
</thead>
<tbody>
<tr>
<td>Human skin</td>
<td>Lactic acid, aliphatic fatty acids, butanal, 3-methylbutanal, 2-methylbutanal, pentanal, hexanal, heptanal, octanal, phenylacetaldehyde, nonanal, decanal, undecanenol</td>
</tr>
<tr>
<td>Foot</td>
<td>Acetic acid, butyric acid, isobutyric acid, isovaleric acid, propionic acid, valeric acid, and isocaproic acid</td>
</tr>
<tr>
<td>Human hair and scalp</td>
<td>Alkanes, alkenes, alcohols, aldehydes, ketones, acids, and 3’-lactone</td>
</tr>
<tr>
<td>Human axillary sweat and sebum</td>
<td>Esters (ethyl-2-methylpropanoate and ethylbutanolate), ketones(1-hexen-3-one and 1-octen-3-one) and, particularly, aldehydes[(Z)-4-heptenal, octanal, (E)-2-octenal, methional, (Z)-2-nonenal, (E,Z)-2,6-nonenadienol, (E,E)-2,4-nonadienol, and 4 methoxybenzaldehyde]</td>
</tr>
<tr>
<td>Breath (mouth air)</td>
<td>Acetone, isoprene, ammonia, ethanol, acetaldehyde, ethylene, trimethyl amine, RSCs (carbonyl sulfide, hydrogen sulfide, methane thiol, and dimethyl sulfide), Saturated VOCs up to C14 + unsaturated C7-C10 + ketononcarboxylic acids (C6-C10) + aldehydes</td>
</tr>
<tr>
<td>Urine</td>
<td>2-butanone, 2-pentanone, 4-heptanone, dimethyl disulfide, alkyl furans, pyrrole, carvone, benzaldehyde, p-cresol, phenol, trimethyl amine, 3-hydroxybutyric acid and acetone</td>
</tr>
</tbody>
</table>

According to the study by Batterman and Peng (2010), the VOCs released from human is in the form of body odour, which is around the rate of 14.8 mg per hour per person. Considering the cabin environments are usually very crowded and relative sealed, this VOCs released rate could be a significant quantity for a long travel period. Since comfort and health issues may happen inside the cabin environment, it is necessary to monitor and control the high concentration of body odour (Wolkoff, 2013, Franklin, 2007).

Due to small sizes of molecule level, the transport behaviours of VOCs are quite different from the other type contaminants in the ventilated indoor environment. Compared with surrounding air, the VOCs molecules have the same physics laws and mathematical equations. For this reason, the molecules of VOCs can easily reach a
human’s lung through respiratory airways. Meanwhile, the VOCs may also have chemical reactions with cells on the mucous membrane which induce the sense of smell. The transport of VOCs in the indoor environment is influenced by both convections with airflow and interaction with surrounding air molecules. More details regarding the calculation of VOCs in HST cabin were presented in Section 3.3.1.

2.1.2 Particle Contaminants

Particulate contaminants are inanimate small grains of mass such as smoke and dust while biological contaminants are animate small grains of mass include pollen, virus, bacteria, mould and pollen. Unlike gaseous contaminants, the sizes of particulate and biological contaminants vary a lot, from 0.001 μm to 1000 μm, which has nearly six orders of magnitude difference (Howie, 1990, Escombe et al., 2007). Relative size chart of common air contaminants has been plotted in Figure 2.1. Under such a large diverse range of size distribution, human’s nasal filtration system is difficult to filter out all the harmful contaminants.

Figure 2.1 Relative size chart of common air contaminants (Howie, 1990)

To identify the critical size range of harmful particulate contaminants, a number of existing studies carefully investigated the particle deposition rate in human’s nasal
cavity (Hsu and Chuang, 2012, Kelly et al., 2004). The deposition efficiency of the inhaled articles measured has been summarised in Figure 2.2 (Kelly et al., 2004, Hsu and Chuang, 2012). It can be noticed from Figure 2.2 that 80% of the particles smaller than 1 nm and larger than 10 μm have been filtered by human’s nasal system. However, for the particle between 100 nm and 5 μm, they have very low deposition efficiency when passing through the nasal system. These range of particles were also identified as PM2.5 (Chen et al., 2016), which are more dangerous than other particles due to their extremely low deposition rate in human's nasal cavity. Meanwhile, a few harmful contaminants are showing in this range such as viruses, bacteria, etc.

Figure 2.2 Particle deposition in nasal cavity

According to the previous study (Yan et al., 2014), the infectious virus released through coughing or sneezing of passengers is the main reason cause disease transmission in public transport. Based on the experiment by Chao et al. (2009), the mean diameter of contaminants from coughing was 13.5 μm with an average release speed of 11.7 m/s. Within half a second, sputum droplets would quickly evaporate into droplet nuclei with an average diameter of 3.5 microns (Redrow et al., 2011). The size evaporated droplet is in the range of PM 2.5 which has low deposit rate in human's nasal cavity. The perniciousness of the contaminants from coughing has attracted many attentions, especially after previous global outbreaks of diseases including SARS and H1N1. Gupta et al. (2011b) numerically and experimentally investigated the distribution of
contaminants released from coughing and breathing. It was found that the contaminants have similar transport characters.

Compared with the gaseous contaminants, particulate contaminants are much larger. Thus, the mathematical equations for gaseous contaminants cannot be applied for particulate contaminants. Li et al. (2014) experimentally measured both gaseous and particulate contaminants transport behaviour. They found that the gaseous contaminants were primarily affected by the airflow, while the particle contaminants could suspend in the air or be carried by indoor airflow, which was affected by more factors. More details regarding the particulate contaminant’s calculation were presented in Section 3.3.2.

In summary of this section, the gaseous contaminants such as body odour (VOCs) and particle contaminants such as droplets released from passengers cannot be neglected in the HST cabin environment. Although the emission rate of VOCs by passengers is in a low concentration, it still can cause the health and smell issues in long time travel. For the particle contaminants, especially pathogen-carrying saliva droplets, it is critical for causing the disease transmission inside public transport.

2.2 HST Ventilation Standard and Thermal Comfort Index

2.2.1 Ventilation Standard of Train Cabin

The train cabin ventilation scheme has been proposed by many railway standards in different countries. Currently, the Standard TB1951-87 (MRPRC, 1987), EN13129-2016 (EU, 2016), UIC553 (UIC, 2005) were widely used around the world. Since the UIC553 was set according to EN13129-2016, most parameters between EN13129-2016 and UIC553 were similar. This study mainly focused on Standard EN13129-2016. Considering the complexity train operation environment, the usage condition for each standard was different. Standard TB1951-87 was used in China, while standard EN13129-2016 was mainly used in Euro countries. In EN13129-2016, three temperature zones have been set, which is Zone 1 for Greece, Great Birth, Portugal and Spain, Zone 3 for Finland, Norway and Sweden, and Zone 2 for the rest Eupen
countries. The train cabin outside temperature parameters for thermal load calculation was listed in Table 2.2.

*Table 2.2 Outside environment parameters for calculation*

<table>
<thead>
<tr>
<th>Standard:</th>
<th>Applied region</th>
<th>Summer</th>
<th>Winter</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td>Exterior temperature for calculation (°C)</td>
<td>Relative humidity</td>
</tr>
<tr>
<td>TB1951-87</td>
<td>China</td>
<td>+35</td>
<td>60%</td>
</tr>
<tr>
<td>EN13129-2016</td>
<td>Zone 1</td>
<td>+40</td>
<td>40%</td>
</tr>
<tr>
<td></td>
<td>Zone 2</td>
<td>+35</td>
<td>50%</td>
</tr>
<tr>
<td></td>
<td>Zone 3</td>
<td>+28</td>
<td>45%</td>
</tr>
</tbody>
</table>

Based on different outside temperature, the cabin interior temperature requirement was also different. For the passengers seating area, the temperature requirements were presented in Table 2.3.

*Table 2.3 Cabin interior temperature standards for the passenger seating area*

<table>
<thead>
<tr>
<th>Standard:</th>
<th>Applied region</th>
<th>Summer °C</th>
<th>Winter °C</th>
</tr>
</thead>
<tbody>
<tr>
<td>TB1951-87</td>
<td>China</td>
<td>24-28</td>
<td>18-20</td>
</tr>
<tr>
<td>EN13129-1-2002</td>
<td>Zone 1</td>
<td>26-28</td>
<td>21-23</td>
</tr>
</tbody>
</table>
The airflow velocity and fresh air flow rate inside the cabin have also been suggested by Standards, as listed in Table 2.4 and Table 2.5.

Table 2.4 Cabin interior airflow velocity standard for the passenger seating area

<table>
<thead>
<tr>
<th>Standard</th>
<th>Applied region</th>
<th>Summer (m/s)</th>
<th>Winter (m/s)</th>
</tr>
</thead>
<tbody>
<tr>
<td>TB1951-87</td>
<td>China</td>
<td>&lt; 0.25</td>
<td>&lt; 0.2</td>
</tr>
<tr>
<td>EN13129-1-2002</td>
<td>Zone 1</td>
<td>0.09~0.6</td>
<td>0.07~0.25</td>
</tr>
<tr>
<td></td>
<td>Zone 2</td>
<td>0.09~0.6</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Zone 3</td>
<td>0.07~0.25</td>
<td></td>
</tr>
</tbody>
</table>

Table 2.5 Cabin fresh air flow rate standard for the passenger seating area

<table>
<thead>
<tr>
<th>Standard</th>
<th>Applied weather</th>
<th>Fresh air flow rate (m³/h/person)</th>
</tr>
</thead>
<tbody>
<tr>
<td>TB1951-87</td>
<td>Summer</td>
<td>20~25</td>
</tr>
<tr>
<td></td>
<td>Winter</td>
<td>15~20</td>
</tr>
<tr>
<td>EN13129-1-2002</td>
<td>Less -20°C</td>
<td>10</td>
</tr>
<tr>
<td></td>
<td>-20~ -5°C</td>
<td>15</td>
</tr>
<tr>
<td></td>
<td>-5~26°C</td>
<td>20</td>
</tr>
</tbody>
</table>
Overall, both Standard TB1951-87 and EN13129-2016 give detail requirements for the interior cabin environment, including thermal comfort, air condition operation, air ventilation scheme, heating and cooling procedures. In order to meet all the requirements from standards, it is essential to investigate and identify the major influencing factors that would significantly optimize the air quality inside HSR cabins.

### 2.2.2 Thermal Comfort Index

According to the American Society of Heating Refrigeration and Air Conditioning Engineers (ASHRAE), the definition of human thermal comfort was expressed as the state of people’s mind that expresses satisfaction with the surrounding environment (ASHRAE, 2004). The thermal comfort depends on many factors. Among these factors, four environmental variables are considered as most important, which are: the air temperature, the mean radiant temperature, the relative air velocity, and the relative humidity.

In recent years, the evaluation of the thermal comfort inside transportation cabins has been studied by many researchers. Currently, there are several indices that have been used for evaluating thermal comfort by international standards. One of these indices is the Predicted Mean Value (PMV) index (Gilani et al., 2015), which predicts the response of the thermal vote by many people in the same environment. Even though the PMV presents the empirical fit to human sensation, it has been developed as the mathematical formulation, which is proposed by Fanger (1972). In Fanger’s mathematical model, the energy balance for a human is applied by using the different energy exchange mechanisms and physiological parameters which are derived from experiment. More details of PMV calculation were discussed in Section 3.5. Table 2.6 describes the thermal sensation scale based on PMV index value.
### Table 2.6 PMV thermal sensation scale

<table>
<thead>
<tr>
<th>Sign</th>
<th>Value</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>+</td>
<td>3</td>
<td>Hot</td>
</tr>
<tr>
<td>+</td>
<td>2</td>
<td>Warm</td>
</tr>
<tr>
<td>+</td>
<td>1</td>
<td>Slightly warm</td>
</tr>
<tr>
<td>0</td>
<td></td>
<td>Neutral</td>
</tr>
<tr>
<td>-</td>
<td>1</td>
<td>Slightly cool</td>
</tr>
<tr>
<td>-</td>
<td>2</td>
<td>Cool</td>
</tr>
<tr>
<td>-</td>
<td>3</td>
<td>Cold</td>
</tr>
</tbody>
</table>

Similar with PMV, the Predicted Percentage Dissatisfied (PPD) is another proposed index, which calculates a prediction of the ratio of thermally dissatisfied people in an environment (Gilani et al., 2015). The PPD is used for predicting the how many people that feel uncomfortable warm or cold in the same environment. It is a quantitative prediction of the percentage of thermally dissatisfied people based on the calculation of PMV index. More details regarding PPD calculation could be found in Section 3.5. Figure 2.3 illustrates the relationship between PMV and PPD. As can see, when the PMV index is near 0, the PPD value is also near 0 correspondingly. With the PMV changes to +3 (hot) or -3 (cold), the PPD values also exponential growth into 100%.
According to ASHRAE (ASHRAE, 2004), draft which is unwanted local cooling due to air movement, can cause occupant dissatisfaction in indoor spaces. The sensation of draft depends on the speed of air, the fluctuating of air velocity (turbulence), the air temperature and clothing conditions. Sensitivity to draft is most likely to be felt on the skin where is not covered by clothing, especially the neck, head, shoulders, ankle, feet and legs. Draft could increase with increasing air speed and turbulence intensity meanwhile could decrease with increasing air temperature (Liu et al., 2017a). Even though both PMV and PPD have considering the temperature, relative velocity and humidity, the influence of turbulence intensity is included in the formula of neither PMV nor PPD. Thus, the percentage of dissatisfied (PD) due to draft model was applied for the thermal comfort analysis (Gilani et al., 2015).

Current thermal comfort standards, such as ASHRAE Standard 55-2004 (ASHRAE, 2004) and ISO 7730 (ISO, 2005), quantify defines draft in terms of what percentage of occupants will be dissatisfied in an environment due to the uncomforting local draft conditions. The model was also proposed by Fanger, which is developed based on the curve-fitting experimental data, and the empirical model of human skin heat transfer. Thus, PD is expressed as a function of convective heat loss. This model relates air speed, temperature, and turbulence intensity to the percentage dissatisfied with air movement around people. More details of PD calculation were discussed in Section 3.5.

Figure 2.3 The PPD index as the relation to PMV index.
According to ASHRAE Standard 55-2004 (ASHRAE, 2004), the PD should be < 20% for a comfort human occupancy environment.

For evaluating the PD value in a specific environment, the measurement locations are to be defined with respect to the occupant (ASHRAE, 2004). Three spatial locations are required to evaluate an occupant’s environment, which are head area, waist area and ankle area. For a seated occupant, which is most common in HST cabin environment, the assessment levels are recommended as 1.1, 0.6 and 0.1 m above the floor.

As mentioned, HST cabins are densely occupied spaces with passengers approximately seated to each other. Passengers are long time exposed in this environment during the trip. Thermal comfort is one of the most important factors for passenger to considering when choosing the transportation. Therefore, it is an important objective for the ventilation system in an HST cabin to create a comfortable thermal environment. Thus, the above-mentioned thermal comfort indexes should be evaluated during the assessment of HST cabin environment.

2.3 Influential Factors of Air Quality in HST cabins

According to Han and Huang (2005), the in-cabin environments is affected by a large number of parameters that include the air temperatures, the air velocity and its profiles over the different geometries, the relative humidity, the solar radiation, the type of ventilation equipment, the response to the change of exterior environment, etc. More importantly, most of these parameters are dependent with unknown relationships. In order to understand the air quality and thermal comfort, these impact factors have been classified as interior cabin and exterior cabin.

2.3.1 Interior Impact Factors

2.3.1.1 Diffuser Design

As one of the crucial components of cabin ventilation systems, diffusers have been widely investigated both experimentally and simulative. There are many types of diffuser applied in HST cabins, as shown in Figure 2.4. Daithankar et al. (Daithankar et
al., 2015) found the diffuser locations can influence passengers’ thermal comfort in a minibus cabin, and improper diffuser orientation angles would lead to a higher temperature for drivers and passengers next to windows. Zhu et al. (2012) simulated four types of ventilation scenarios of a bus cabin including using linear diffusers and round diffusers. They found that diffuser type, filtration of recirculated air, location of exhaust vents and arrangements of passenger seats could all influence airborne diseases transmission. Chow (2002) studied the air-distribution performance in an enclosed train cabin. Their results indicated that the air diffusion terminal devices must be located at the upper level so that carbon-dioxide level in crowded train cabins can be well controlled at a low level. Zhang and Li (2012) found that in HST cabins, the back-door-oriented design of exhaust outlet coupled with the CRH3 model cabin inlet diffuser has strong ability to remove cough droplets. Wang et al. (2014) studied ventilation performance in three HST cabins including the CRH1, CRH2, and CRH5 models, which have similar cabin size and seat arrangements but different ventilation schemes. The results revealed that CRH5 has the best ventilation efficiency. Li et al. (2014) reported that on-the-market mainstream air diffusers in airliner cabins can be categorised as personal gaspers (on each passenger overhead), wall grilles (at the upper of the sidewalls) and roof grilles (at the top of aisle). These three types of diffuser have different ventilation characteristics regarding to different cabins and working conditions. Study also found that the mixed application of air diffusers would exert the best ventilation performance for the cabin environment (Yan et al., 2015).

In order to minimize disease transmission and improve ride comfort, most modern HST cabins are ventilated using an up-to-down air-conditioning scheme. This type of ventilation design sets inlets near the cabin ceiling and outlets near the floor, creating a descending overall airflow pattern which helps retain contaminants below passengers’ breathing zone. However, due to the metabolic heat released by passengers, a strong upward buoyancy flow is also created in cabins, which competes against the ventilating air flow and reduces the effectiveness of contaminant removal. In order to improve the effectiveness of contaminant control, a large flow rate of supplement fresh air is recommended (Zhuang et al., 2014), which however, would largely increase the energy consumption and degrade passengers’ thermal comfort. Therefore, optimized
ventilation schemes, particularly carefully designed ventilation diffusers, are necessary to be developed to minimize disease transmission and improve the air quality in public transportation vehicle cabins while maintaining the ride comfort and reducing the energy consumption (Zhang et al., 2009a).

**Figure 2.4 Popular types of cabin diffuser design**

#### 2.3.1.2 Passenger Activities – Coughing Process

The passengers, who are the occupants of HSR cabin, are the most important releasing sources of indoor contaminants. Batterman and Peng (2010) found an average adult person in rest state would emit 14.8mg VOCs (body odour) per hour. Meanwhile, some particle contaminants such as the infectious virus are released through coughing or sneezing of passengers, which can cause the disease transmission as discussed in Section 2.1.2. Thus, passenger activity is one important influential factor of air quality inside the cabin.
When passengers were coughing, one of the factors that released simultaneously with the contaminants through the coughing process, the cough-jet, was overlooked in current studies. Most existing studies assumed the cough-jet were ignorable, as the cough-jet could be quickly damped after release (Chen et al., 2013). For instance, Li et al. (2016b) numerically investigated the airborne contaminant distribution realised from one manikin’s mouth in a 7-row airliner cabin section. In their study, only contaminants were released though coughing without generating any cough-jet. This was also because it would require a significantly high computational cost to investigate the cough-jet characteristics in a full-scale airliner cabin due to the extreme complexity of the cabin environment. According to Gupta et al. (2011b), tracking coughing droplets transport for 4 minutes in a 7-row cabin took more than 4 weeks to finish using a fully operated computer cluster.

On the other hand, the cough-jet even in the first few seconds was argued as an important factor on the local airflow distribution from other indoor spaces (Yang et al., 2017). Kwon et al. (2012) pointed out that an accurate description of the initial velocity distribution from coughing process is particularly important in the study of the fluid dynamics property of the respiratory particles. Their study concluded that only considering the airflow field without the cough-jet may not be an appropriate representation of the transport and distribution of airborne contaminants. By experimental studying, the concentration of contaminants in aircraft cabin during several long-distance flights, Dechow et al. (1997) illustrated that the peak concentrations of bacteria can be significantly observed after coughing from the passenger. Li et al. (2016b) set the experiment for study the cabin environment (Figure 2.5 up-left and up right), and Gupta et al. (2009) experimentally studied the coughing character as shown in Figure 2.5 (down-left and down right). Gupta et al. (2011a) also argued that the cough-jet driven by the coughing process should be considered as one of the most important factors in airborne disease transmission. In cabin environment, due to the high densely, cough-jet released by source passengers could break the local airflow not only in their but also their neighbours’ breathing zone. Thus, it is crucial to include the cough-jet when investigating the contaminants transport and distributions through coughing in the cabin environment.
2.3.2 Exterior Impact Factors

2.3.2.1 HST Operation Conditions

During the past decades, High-speed trains (HSTs) are rapidly developed in many countries with the operation speed increased from 200 km/h to 500 km/h (Liu et al., 2016). The speed-up train system has attracted many researchers’ interests on aerodynamic problems due to the impact of strong disturbance on the surrounding air (Zhao et al., 2013). Particularly, when two HSTs are passing each other and HST is passing through the tunnels, the disturbance will be aggravated, which would induce very large transient pressure fluctuations, as see in Figure 2.6. Liu et al. (2016)’s research found that the pressure wave on the cabin outside surface could influence running security and passenger comfort. Zhao et al. (2013) studied the HSTs passing behaviours at different speed levels and found that pressure wave was proportional to
the train speed. Raghunathan et al. (2002) also demonstrated that the induced pressure wave was one of the most important limitations to HST speed-up.

Most recent studies of induced pressure wave focus on the pressure wave acting on the outside surface of HST such as the aerodynamic and aeroacoustic performances. However, according to Liu and Li (2012), the induced pressure wave could also influence the cabin inside air environment through the cabin ventilation system. The exhaust vents of air conditioners were exposed to the outside surface of train cabin. As the cabin inside air environment was mainly dominated by the air conditioning system (Xu et al., 2013), the induced pressure wave could indirectly affect the cabin inside air environment. In addition, Zhang and Li (2012) found that the outside pressure could also affect the inside cabin environment through the cabin connection gaps. Their research found that the back-door-gap-oriented design of exhaust outlets in the CRH3 model cabin had strong ability in removing cough droplets.

At the meantime, HST cabins are densely occupied spaces with passengers approximately seated to each other. Huruya (Furuya, 2007) warned that train cabins could be a high-risk venue for the airborne transmission of infectious diseases such as Tuberculosis and influenza. However, the transport and dispersion characteristics of airborne contaminants in fluctuating pressure fields have been rarely studied. As the extensive studies on public transport cabins (Liu and Chen, 2013, Zhang et al., 2009b) have proven that the dispersion of airborne contaminants in enclosed cabin spaces is predominantly controlled by the airflow field, a thorough understanding of the effects of pressure fluctuations on the thermal flow field and contaminant dispersion in HST cabins is crucial to create a comfort and safe riding environment. Unfortunately, the important issue has long been overlooked. Therefore, it is necessary to achieve a quantitative analysis of the effects of the pressure fluctuations on the thermal environment and contaminant transport characteristics in HST cabins.
Figure 2.6 Exterior pressure waves when HST entering a tunnel (Yang, 2018)

2.3.2.2 Solar Radiation

According to a study by Simion et al. (2016), solar radiation is one of the crucial issues in ventilation system design. It can significantly affect ventilation system performance in an enclosed space. As shown in Figure 2.7, the thermal balance for the cabin environment is directly affected by solar radiation. In the past decade, a number of simulative and experimental studies have investigated the effect of solar radiation on HVAC and thermal comfort in the vehicular environment. By comparing the simulative and experimental results, Neacsu et al. (2017) found that the sun position has a strong influence on cabin temperature and passenger’s comfort. Lee et al. (2014) pointed out that the solar radiation could heat up both exterior and interior surface of a car and consequently change the thermal comfort. Based on different simulative results of numerical radiation models, Moon et al. (2016) demonstrated that the spectral radiation effect must be considered in CFD simulation for accurate car cabin air temperature prediction. A 1-2°C temperature increment has been found after involving the radiation
model in the simulation. All these studies revealed the fact that solar radiation plays important roles in ventilation performance and passengers’ thermal comfort in passenger compartments.

However, studies on the effect of solar radiation on thermal comfort in HST cabins are rare to see. Unlike metros which can be operated in tunnels or on shade ways, HSTs are usually operated in the open air where they are directly exposed to the solar load (Raghunathan et al., 2002). Thus, HSTs are more likely to suffer solar radiation. Additionally, the process of radiative rays transmitting through windows into HST cabin is complex, which involved many impact factors. On one hand, the solar radiation condition is based on a dynamic environment, which related to solar emitting angle, radiation intensity, local time, location coordinates, and weather condition (Ahmad and Tiwari, 2011, Cook et al., 2008). On the other hand, the HST cabin itself could affect the results of solar radiation, such as the train travelling direction, the material of cabin and window, the arrangement of passengers, and cabin geometry (Yang et al., 2017, Bouvard et al., 2018). Therefore, considering the specific characters of HST cabin and the complexity of solar radiation process, it is necessary to access the thermal comfort in HST cabins under solar radiation environment.

Figure 2.7 Solar radiation on the HST cabin (up-left); Solar radiation through the HST cabin window (up-right); Thermal balance of HST cabin when considering solar radiation (down)
2.4 Optimization algorithm coupled with CFD

To obtain the optimal design, in the conventional “trial and error” design cycle, system design parameters such as supply airflow rate, air temperature and humidity are manually adjusted and evaluated based on on-site measurements or analytical and empirical models (Liu et al., 2015). Over the past decades, computational fluid dynamics (CFD) techniques have been widely adopted to predict the indoor air environment aiming to shorten the time and reduce the cost of the lengthy HVAC system design cycles in buildings (Ravikumar and Prakash, 2009, Cardinale et al., 2010, Hiyama et al., 2010, Kochetov et al., 2015, Limane et al., 2018, Tian et al., 2018, Cao et al., 2014, Pu et al., 2014, Wang and Zhai, 2016, Yan et al., 2016) and long-haul transportation cabins (Liu et al., 2012a, Li et al., 2016b, Liu et al., 2013, Kwon et al., 2009, Zhang and Li, 2012, Konstantinov and Wagner, 2014, Konstantinov and Wagner, 2015). With no doubt, compared to the on-site measurement or experimental analysis, the CFD technique appears to be a cost and time effective tool for design optimization. Nevertheless, due to the nature of trial and error design process, a large number of simulations is usually required for covering the entire design space; leading to significant computational time and resource.

In order to reduce the computational time, some surrogate techniques such as artificial neural network (ANN) (Zhou and Haghighat, 2009a, Zhou and Haghighat, 2009b, Acikgoz et al., 2017, Bre et al., 2018), Support Vector Machine (SVM) (Mousa et al., 2017), Kriging (Li et al., 2017) and Proper Orthogonal Decomposition (POD) (Li et al., 2013) are employed as a fast alternative approach replacing the CFD simulation to approximate the nonlinear and complex behaviour of the indoor airflow. On the other hand, aiming to automate the trial and error process, evolutionary algorithms (EA) have been proposed and coupled with the CFD to search the globally optimal solution (Luh and Lin, 2011, Li et al., 2013, Zhai et al., 2014). Although the CFD-EA coupled approach significantly reduces the required number of CFD simulations to reach optimal solution, it still requires a substantial amount of CFD simulations for training the surrogate models to construct a reliable response space for EA.
Furthermore, the evaluation of indoor environment can be a complex process where multiple evaluation criteria are normally involved such as air velocity and temperature near the occupants, contaminant concentrations, percentage dissatisfied of draft, age of air, total energy consumption, etc. The system design is therefore a multi-objective optimization process where trade-off relations among those design indices are usually needed to be considered. Especially, in terms of indoor thermal comfort evaluation, lots of research works have been done by Ricciardi’s group (Buratti and Ricciardi, 2009, Buratti et al., 2013, Nematchoua et al., 2014, Ricciardi and Buratti, 2015, Buratti et al., 2016, Ricciardi et al., 2016). In most of the previous works, in order to handle a multi-objective problem, all the design objectives are aggregated into one single objective function through pre-defined biased weighting factors (Laverge and Janssens, 2013, Li et al., 2013). One of the major disadvantages of this method is that the optimal design could be sensitive to the weighting factors, thus different values of the weights could result in substantially different solutions. Therefore, the values of these weighting factors are objected to professional knowledge and expert judgements. In addition, the optimization procedure provides only one optimal solution per simulation run, giving the designers no flexibility in selecting alternative solutions for striking a trade-off of the conflicting parameters.

In attempting to address the above shortcomings for the design of HST cabin ventilation system, it is necessary to propose a novel design scheme where a nondominated sorting-based particle swarm optimization (NSPSO) algorithm is utilized to achieve multi-objective optimization without using any biased weights.
Chapter 3
Methodology

3.1 Project Breakdown

As mentioned in Section 2.3, the air quality in HSR cabin is easily affected by many influential factors from both inside and outside the cabin. More importantly, most of these influential factors are dependent with unknown relationships. It is extremely difficult to optimize air quality by considering all of these influential factors. The numerical outcomes are less robust if simulations are directly conducted under HSR cabin environment without evaluating each influential factor of the cabin environment. Thus, the strategy of this research is to first break down the whole project into multiple components. Then, the investigations of each major affecting factor are conducted to obtain the fundamental knowledge for conducting the optimization purpose. The whole project breakdown process is illustrated in Figure 3.1

This evaluation research started with the study of interior impact factors. The diffuser type is an important part of cabin design and it is key to the HST cabin ventilation performance. Four different types of diffusers which are widely used in currently HSTs were investigated. Then, another interior impact fact, the passengers cough process, was analysed to study the particle contaminant transport characteristics. Moreover, the study on exterior impact factors focused on both special operation conditions and solar radiation efforts. The response of cabin interior airflow, contaminant transport and
thermal comforts to the transient pressure fluctuation during the period of two HSTs passing by each other were studied at first. Then, the study of another operation condition, the HST passing through a tunnel, was carried out. Lastly, the interior airflow and thermal comfort responding to the radiations of different daytime and window conditions were also analyzed.

Based on the investigated influential factors, a multi-objective algorithm model was proposed to optimise HST cabin ventilation system and achieve better computational efficiency. With the optimization algorithm, different combinations of ventilation operation parameters were investigated against the ventilation performance in terms of thermal comfort, air quality and energy consumption. Eventually, with the multi-objective algorithm, all the tested components can be integrated to establish a platform for future investigations under similarly densely-occupied environments.

![Diagram of Interior and Exterior Impacts](image)

**Figure 3.1 Schematic view of breaking down the project into separate components**

### 3.2 Governing Equations for Airflow

Although the HSR cabin is a complex and multi-scale environment, it is still considered as a continuum in CFD simulation. The behaviours of airflow in relation to velocity,
pressure, temperature and density is described using the incompressible Navier-Stokes (NS) equation, which is also known as the governing equations.

The governing equation of the airflow is obtained by identifying the fundamental principles based on conservation of mass, Newton's second law for the conservation of momentum and the first law of thermodynamics for the conservation of energy. It can be expressed as follows:

\[
\frac{\partial \phi}{\partial t} + \nabla \cdot (\phi \vec{V}) = \nabla \cdot (\Gamma \nabla \phi) + S_\phi
\] 3.1

In the Cartesian coordinate system, it can be expressed as:

\[
\frac{\partial \phi}{\partial t} + \frac{\partial (\phi u)}{\partial x} + \frac{\partial (\phi v)}{\partial y} + \frac{\partial (\phi w)}{\partial z} = \frac{\partial}{\partial x} (\Gamma \frac{\partial \phi}{\partial x}) + \frac{\partial}{\partial y} (\Gamma \frac{\partial \phi}{\partial y}) + \frac{\partial}{\partial z} (\Gamma \frac{\partial \phi}{\partial z}) + S_\phi
\] 3.2

Where \( \phi \) is the general fluid variable; \( t \) is the time; \( \vec{V} \) is the fluid velocity; \( u, v, w \) are the local velocity components of \( \vec{V} \); \( \Gamma \) is the general diffusion coefficient; \( S_\phi \) is the source term. The items in this equation represent the change rate of \( \phi \), convective of \( \phi \), diffusion of \( \phi \), and source term of \( \phi \), respectively.

**3.2.1 Continuity Equations**

The continuity equations are based on the conservation of mass. The scalar variable \( \phi \) is identified as the fluid density \( \rho \) in the governing equation, while the diffusion term is discarded. The continuity equations can be written as:

\[
\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \vec{V}) = 0
\] 3.3

In the Cartesian coordinate system, it can be expressed as:

\[
\frac{\partial \rho}{\partial t} + \frac{\partial (\rho u)}{\partial x} + \frac{\partial (\rho v)}{\partial y} + \frac{\partial (\rho w)}{\partial z} = 0
\] 3.4
3.2.2 Momentum Equations

The momentum equations are identified based on Newton’s second law. In the governing equation, the scalar variable $\Phi$ is identified as the fluid velocity $u$, while the diffusion term $\Gamma$ is replaced by the viscosity $\mu$. The force is presented as pressure $p$, which is added in the source term. The conservation form of the momentum equation can be rewritten from the conservation equations as:

$$\frac{\partial \vec{V}}{\partial t} + (\vec{V} \cdot \nabla) \vec{V} = \nu \nabla^2 \vec{V} - \frac{1}{\rho} \nabla p$$

In the Cartesian coordinate system, it can be expressed as:

X momentum:

$$\frac{\partial u}{\partial t} + u \frac{\partial u}{\partial x} + v \frac{\partial u}{\partial y} + w \frac{\partial u}{\partial z} = \mu \left( \frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2} + \frac{\partial^2 u}{\partial z^2} \right) - \frac{1}{\rho} \frac{\partial p}{\partial x}$$

Y momentum:

$$\frac{\partial v}{\partial t} + u \frac{\partial v}{\partial x} + v \frac{\partial v}{\partial y} + w \frac{\partial v}{\partial z} = \mu \left( \frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2} + \frac{\partial^2 v}{\partial z^2} \right) - \frac{1}{\rho} \frac{\partial p}{\partial y}$$

Z momentum:

$$\frac{\partial w}{\partial t} + u \frac{\partial w}{\partial x} + v \frac{\partial w}{\partial y} + w \frac{\partial w}{\partial z} = \mu \left( \frac{\partial^2 w}{\partial x^2} + \frac{\partial^2 w}{\partial y^2} + \frac{\partial^2 w}{\partial z^2} \right) - \frac{1}{\rho} \frac{\partial p}{\partial z}$$

3.2.3 Energy Equation

The momentum equations are identified based on the first law of thermodynamics for the conservation of energy. In the governing equation, the scalar variable $\Phi$ is identified as the fluid temperature $T$, while the diffusion term $\Gamma$ is replaced by the thermal diffusivity ($\alpha = k/\rho C_p$).
\[
\frac{\partial T}{\partial t} + (\mathbf{V} \cdot \nabla)T = \alpha \cdot \nabla^2 T + S_T \tag{3.9}
\]

In the Cartesian coordinate system, it can be expressed as:

\[
\frac{\partial T}{\partial t} + u \frac{\partial T}{\partial x} + v \frac{\partial T}{\partial y} + w \frac{\partial T}{\partial z} = \frac{\lambda}{\rho C_p} \left( \frac{\partial^2 T}{\partial x^2} + \frac{\partial^2 T}{\partial y^2} + \frac{\partial^2 T}{\partial z^2} \right) + S_T \tag{3.10}
\]

where, \( \lambda \) is the thermal conductivity, \( C_p \) is the thermal capacity and \( S_T \) is the internal thermal source.

### 3.2.4 Turbulence Modelling

Turbulent flow is the irregular state of fluid motion characterized by chaotic changes in pressure and flow velocity. In currently CFD simulations, the turbulent flow can be solved using the direct numerical solution (DNS), the Large-eddy simulation (LES) or the Reynolds-Averaged Navier-Stokes (RANS) model. The DNS method resolves the all ranges of the turbulence in both spatial and temporal scales, while the LES model resolves all the large eddies. Therefore, when using DNS and LES methods, detailed turbulent eddy information could be obtained, meanwhile significantly high computational cost is also required. The RANS model approximates time-averaged solutions to the Navier–Stokes equations by decomposing the instantaneous quantities, while still provides reasonably reliable solutions to turbulence flows with less computational cost and time. Thus, RANS is suitable for a very complex computational domain such as HST cabin environment.

In the RANS model, the scalar variable \( \varnothing(t) \) in the instantaneous Navier-Stokes equations is decomposed into the mean component \( \overline{\varnothing} \) and the fluctuating component \( \varnothing'(t) \):

\[
\varnothing(t) = \overline{\varnothing} + \varnothing'(t) \tag{3.11}
\]

The velocity components can be expressed as:
\[ u(t) = \bar{u} + u'(t) \]

Where \( \bar{u} \) and \( u'(t) \) are the mean and fluctuating velocity components of \( u(t) \). The intensity of the turbulence flow, \( I \), can be defined by the ratio of the fluctuating velocity to the mean velocity as:

\[ I = \frac{u'}{\bar{u}} = \frac{\sqrt{\frac{1}{3}(u_x'^2 + u_y'^2 + u_z'^2)}}{\sqrt{\frac{1}{3}(\bar{u}_x^2 + \bar{u}_y^2 + \bar{u}_z^2)}} \]

In which the mean velocity \( \bar{u} \) is:

\[ \bar{u} = \frac{1}{t} \int_{t_0}^{t_0+t} u(x,y,z) dt \]

And the mean and fluctuating velocity component is:

\[ \bar{u}'(t) = \frac{1}{t} \int_{t_0}^{t_0+t} u'(t) dt = \frac{1}{t} \int_{t_0}^{t_0+t} (u(t) - \bar{u}) dt = 0 \]

By substituting expressions of the decomposed form for the flow variables into the continuity equation and taking a time average, the mass conservation equation can be rewritten as:

\[ \frac{\partial \bar{u}}{\partial x} = 0 \]

Similarly, the momentum equation can be rewritten as:

\[ \frac{\partial \bar{u}_i}{\partial t} + \frac{\partial}{\partial x} (\bar{u}_i \bar{u}_j) = v \frac{\partial^2 \bar{u}_i}{\partial x^2} - \frac{1}{\rho} \frac{\partial p}{\partial x} + \frac{\partial}{\partial x} (-\bar{u}_i' \bar{u}_j') \]

The above equations are called Reynolds-averaged Navier-Stokes (RANS) equations. The additional term \( \bar{u}_i' \bar{u}_j' \) representing the effects of turbulence is related to the Reynolds stress tensor \( R_{ij} \):
\[ R_{ij} = \rho \overline{u_i'u_j'} \]

### 3.2.5 The RNG k-epsilon Turbulence Model

For the air turbulence in enclosed complexity spaces, such as cabin environment, the RNG (Re-Normalisation Group) \( k - \varepsilon \) model has been widely employed due to its good performance on modelling indoor airflow and pollutant transport (Chen and Chao, 2016). The RNG \( k - \varepsilon \) model was developed from the standard \( k - \varepsilon \) model to account for the effects of smaller scales of motion by using a modified form of the epsilon equation through changes to the production term (Yakhot and Orszag, 1986, Yakhot et al., 1992). In the standard \( k - \varepsilon \) model (Launder and Spalding, 1974) in which \( k \) represents the turbulence kinetic energy and \( \varepsilon \) stands for its rate of dissipation:

\[
k = \frac{1}{2} \overline{u'_i u'_l} \quad \text{(3.19)}
\]

\[
\varepsilon = \nu \left( \frac{\partial u'_i \partial u'_l}{\partial x_j \partial x_j} \right) \quad \text{(3.20)}
\]

Since the standard \( k - \varepsilon \) model is only applicable for fully turbulent flows (Chen, 1995b), it requires very comprehensive wall functions to provide accurate prediction of air turbulence. Although transport coefficients for the walls are obtained through experimental measurements, they are not universal (Chen, 1995b). Yakhot and Orszag (1986) using a statistical technique so-called renormalization group (RNG) methods to develop a theory for the large scales, in which the effects of the small scales are represented by modified transport coefficients. Thus, the RNG \( k - \varepsilon \) model has a broader applicability and is more reliable and accurate than the standard \( k - \varepsilon \) model under very complex environment. The transport equation of the RNG \( k - \varepsilon \) model has a similar form to the standard \( k - \varepsilon \) model, and in the ANSYS is given as:
\[
\frac{\partial (pk)}{\partial t} + \frac{\partial}{\partial x_i} (pk u_i) = \frac{\partial}{\partial x_j} \left( \alpha_k \mu_{\text{eff}} \frac{\partial k}{\partial x_j} \right) + G_k + G_b - \rho \varepsilon - Y_M + S_k \tag{3.21}
\]

\[
\frac{\partial (\rho \varepsilon)}{\partial t} + \frac{\partial}{\partial x_i} (\rho \varepsilon u_i) = \frac{\partial}{\partial x_j} \left( \alpha_\varepsilon \mu_{\text{eff}} \frac{\partial \varepsilon}{\partial x_j} \right) + C_{1\varepsilon} \frac{\varepsilon}{k} (G_k + C_{3\varepsilon} G_b) - C_{2\varepsilon} \rho \frac{\varepsilon^2}{k} - R_\varepsilon + S_\varepsilon \tag{3.22}
\]

\[\mu_{\text{eff}} = \mu + \mu_t \tag{3.23}\]

\[\mu_t = \rho C_\mu \frac{\varepsilon^2}{k} \tag{3.24}\]

Where, \(G_k\) represents the generation of turbulent kinetic energy due to the mean velocity gradients, and \(G_b\) is the generation of turbulent kinetic energy due to buoyancy. \(\mu_{\text{eff}}\) is the effective viscosity, and \(\mu_t\) is the turbulent viscosity. The quantities \(\alpha_k\) and \(\alpha_\varepsilon\) are the inverse effective Prandtl numbers \(Pr\) for \(k\) and \(\varepsilon\), respectively, \(\alpha_k = \alpha_\varepsilon \approx 1.393\). The model constants values \(C_{1\varepsilon} = 0.0845\), \(C_{2\varepsilon} = 1.42\) and \(C_{3\varepsilon} = 1.68\) are derived based on the analytically of the RNG theory.

### 3.3 Governing Equations for Contaminants

#### 3.3.1 The Drift-flux Model for Gaseous Contaminants

As mentioned before, the gaseous contaminants studied in this project mainly focused on the VOCs released from passengers. Due to the low concentration of VOCs inside the HSR cabin (as discussed in Section 2.1.1), the transportation process of gaseous contaminants was modelled using the drift-flux model:

\[
\frac{\partial \varphi}{\partial t} + \frac{\partial}{\partial x_i} (U \varphi) = \frac{\partial}{\partial x_i} \left( D_\varphi \frac{\partial \varphi}{\partial x_i} \right) + S_\varphi \tag{3.25}
\]

where \(U\) is the fluid velocity in air-flow domain; \(\varphi\) is the concentration of gaseous contaminant; \(S_\varphi\) is the volumetric source term of contaminant; \(D_\varphi\) is the kinematic diffusivity of contaminant through the air.
3.3.2 The Eulerian-Lagrangian Model for Particulate Contaminants

When simulating the particulate contaminants distribution in the HSR cabin, the Eulerian-Lagrangian model is employed, which is the most popular two-phase flow model for modelling particle transport when the dispersed phase occupies a low volume fraction. In this model, the airflow is treated as a continuum by solving the Navier-Stokes equations, while the particulate contaminants were tracked using the Lagrangian approach through the calculated flow field separately. The dispersed phase can exchange momentum, mass, and energy with the fluid phase. The particles tracked in Lagrangian model is based on the equation of motion for each particle. For micron particles immersed in continuous air, important forces governing particle motions are the drag force $F_D$ and buoyancy force $F_B$ (Li et al., 2012b), thus,

$$m_p \frac{dU_p}{dt} = F_D + F_B \tag{3.26}$$

$$F_D = \frac{1}{2} C_D \rho_F A_F |U_S| U_S \tag{3.27}$$

$$F_B = \frac{\pi}{6} d_p^3 (\rho_p - \rho_F) g \tag{3.28}$$

$$U_S = U_p - U_f \tag{3.29}$$

Where $m_p$ is the mass of particle, $U_p$ is the particle velocity, $C_D$ is the drag coefficient, $\rho_F$ is the fluid density, $A_F$ is the effective particle cross section, $U_S$ is the velocity difference between fluid and particle, $d_p$ is the particle diameter, $g$ is the gravity vector.

The effect of turbulent dispersion on particle transport is modelled in the Lagrangian model by adding an eddy fluctuating component onto the mean fluid velocity. It is the fluctuating component of the fluid velocity which causes the dispersion of particles in the turbulent flow. Thus, the instantaneous fluid velocity $U_f$ is decomposed into mean velocity $\bar{U}_f$, and fluctuating eddy velocity $U'_f$ components,
\[ U_f = \overline{U_f} + U'_f \]  

3.30

Considering that most surface area of HSR cabin walls, seats, and passengers’ clothes are made of fabrics, droplets could not bounce back to the fluid domain after collisions. Therefore, the “Stick-to-Wall” model (ANSYS, 2016) was used in the simulation for this phenomenon, which treats these surfaces as a sink term of the particulate phase.

### 3.4 Radiation Model

To model the radiative heat transfer, the HST cabin was considered as an enclosure of grey-diffuse surfaces. The radiative exchanges between these surfaces were calculated using the Surface-to-Surface (S2S) model (ANSYS, 2016). In this model, the energy flux leaving a given surface is composed of direct emitted and reflected energy:

\[ q_{\text{out},k} = \varepsilon_k \sigma T_k^4 + \rho_k q_{\text{in},k} \]  

3.31

where, \( q_{\text{out},k} \) is the energy flux leaving the surface \( k \), \( \varepsilon_k \) is the emissivity of surface \( k \), \( \sigma \) is the Stefan-Boltzmann constant, \( T_k^4 \) is the temperature of surface \( k \), \( \rho_k \) is the density of surface \( k \), and \( q_{\text{in},k} \) is the energy flux incident on the surface \( k \) from surroundings.

When using S2S model, the amount of incident energy upon the surface \( k \) from another surface \( j \) depends on the surface size, separation distances, and orientations. These parameters are accounted by a geometric function, the view-factor \( F_{kj} \), which is the fraction of energy leaving surface \( j \) that is incident on surface \( k \). Thus, the energy exchange between two surfaces \( j \) and \( k \) is expressed as

\[ q_{\text{in},k} = \sum_{j=1}^{N} F_{kj} q_{\text{out},j} \]  

3.32

In this equation, the view-factor is calculated using the integration method (ANSYS, 2016). For randomly selected two surfaces \( j \) and \( k \) in 3.32, the view-factor \( F_{kj} \) can be calculated by
\[ F_{kj} = \frac{1}{A_k} \int_{A_k} \int_{A_j} \frac{\cos \theta_k \cos \theta_j}{\pi r^2} \delta_{kj} dA_j dA_k \]  

where, the parameter \( \delta_{kj} = 1 \) when \( dA_j \) is visible to \( dA_k \) and otherwise \( \delta_{kj} = 0 \). Here, \( A_k \) presents the area of surface \( k \). As the geometry in HST cabin is complex and the total mesh number achieved 8.6million, the integration method for radiation calculation process can be computationally very costly. To reduce the computational cost, the Surface-Clustering technique (ANSYS, 2016) was employed, which, instead of calculating each surface grid element, calculates the mean view factor and the surface temperature of a cluster of surface grid elements.

To account for the radiative effects on the thermal flow field, the radiative heat transfer was jointly solved with the airflow field. To do this, the incident heat flux (Eq.3.31) was incorporated in the CFD model as a boundary condition of the energy equation. The local energy balance of the grid element \( k \) on the solid surface was expressed by

\[ q_{ext} + q_{in,k} - (1 - \epsilon_k)q_{in,k} - h_k(T_k - T_a) - \epsilon_k \sigma T_k^4 = 0 \]  

where, \( q_{ext} \) is the flux of externally applied heat, \( h_k \) is the convective heat transfer coefficient and \( T_a \) is the air temperature. The items on the left-hand side of Eq. 3.35 represents the externally applied, incident, reflected, convective and directly emitted heat flux components, respectively.

### 3.5 Thermal Comfort Index

According to ASHRAE Standard 55 (ASHRAE, 2004), human thermal comfort is the condition of the mind that expresses one’s satisfaction with the thermal environment. To achieve such condition, the metabolic heat generated from a human body should equal to the heat loss from the body. It means a constant heat exchange between a human body and the environment should be maintained. Therefore, an important objective of the ventilation system in an HST cabin is to create a comfortable thermal environment.
For each passenger, comfort variables include air velocity, air temperature, radiant temperature, relative humidity, and turbulence intensity in the occupied zone (ASHRAE, 2004). In order to quantify the ventilation performance, the Predicted Mean Vote (PMV), which is accepted as the most recognized criteria index in evaluating the combination of thermal comfort variables (Lin et al., 2005), was calculated based on the Fanger’s heat balance equation (ASHRAE, 2004).

\[
PMV = [0.303 \exp(-0.036M) + 0.028] \times ((M - W) - 3.05 \times 10^{-5} \times [5733 - 6.99(M - W) - P_a] - 0.42 \times [(m - w) - 58.15] - 1.7 \times 10^{-5} \times M \times (5867 - P_a) - 0.0014M \times (34 - T) - 3.96 \times 10^{-8} \times f_{cl} \times [(T_{cl} + 273)^4 - (T_r + 273)^4] + f_{cl} \times h_c \times (T_{cl} - T))
\]

The clothing surface temperature \(T_{cl}\) in the above equation is determined by

\[
T_{cl} = 35.7 - 0.028(M - W) - I_{cl}(3.96 \times 10^{-8} \times f_{cl} \times [(T_{cl} + 273)^4 - (T_r + 273)^4] + f_{cl} \times h_c \times (T_{cl} - T))
\]

The convective heat transfer coefficient \(h_c\) is determined by

\[
h_c = 2.38(T_{cl} - T)^{0.25} \text{ for } 2.38(T_{cl} - T)^{0.25} \geq 12.1u^{0.5}
\]

\[
h_c = 12.1u^{0.5} \text{ for } 2.38(T_{cl} - T)^{0.25} < 12.1u^{0.5}
\]

The clothing surface area factor \(f_{cl}\) is determined by

\[
f_{cl} = 1.05 + 0.645 \times l_{cl} \text{ for } l_{cl} \geq 0.078
\]

\[
f_{cl} = 1.00 + 1.290 \times l_{cl} \text{ for } l_{cl} < 0.078
\]

Due to the fact that there is a large variation of people’s perception of ambient thermal comfort, an evaluation criterion is set up according to the ISO 7730 (ISO, 2005). More details about the PMV criterion have been discussed in Section 2.2.2.
Based on the value of PMV, the prediction of the percentage of dissatisfaction (PPD) of the thermal comfort was calculated:

$$PPD = 100 - 95 e^{-0.03353PMV^4 - 0.2179PMV^2}$$ \hspace{1cm} (3.41)

However, the influence of turbulence intensity is included in the formula of neither PMV nor PPD. In fact, the turbulence of airflow has a significant impact on thermal comfort. Thus, the percentage of dissatisfied (PD) people due to draft model was also applied for the thermal comfort analysis, expressed as follows:

$$PD = (34 - T)(U - 0.05)^{0.62} (3.14 + 0.37u \times Tu)(\%)$$ \hspace{1cm} (3.42)

$$Tu = 100 \times (2k)^{0.5}/u (\%)$$ \hspace{1cm} (3.43)

For calculating the PMV, PPD and PD in HST cabin, the above equations were implemented on top of the normal CFD program executions.
Chapter 4

The Effects of Diffuser Type on Ventilation and Contaminant Transport in HSR Cabins

The main findings of this chapter have been included in:


Recalling the previous global outbreaks of diseases including SARS and H1N1 spread in public transportation cabins, the investigation of transport characteristics of gaseous and particulate containment through the high-speed trains (HSTs) cabin air is critical due to the currently fast development of high-speed rail (HSR) around the world. For the aim of quantitatively analyzing the effects of diffuser type on ventilation and spread of diseases in HST cabins, CFD models containing four different diffuser types were developed in this study. CFD computations were conducted to assess the effect of diffusers. The drift-flux model and Lagrangian model were applied. Simulation results indicated that the diffuser type significantly affects overall cabin airflow pattern. Even
all four types of diffusers applied in HST cabins could satisfy the thermal comfort criterion, containments dispersion processes varied. Besides, gaseous contaminants and particles contaminants transport also varied in the same ventilation conditions. Importantly, contaminants had different high-concentration zones around passengers due to the uprising passengers’ thermal plume against the descending ventilation airflow. This phenomenon would increase passenger exposure risk.

4.1 Introduction

High-speed trains (HSTs) play an important role in domestic and international passenger transportation owing to their unparalleled transport capacity and high efficiency. Inspired by the success of French TGV, Japanese Shinkansen and recent Chinese Railway High-speed Train (CHR), an increasing number of countries including America and Australia have been developing, or planning to develop, their own high-speed rail (HSR) systems (AECOM, 2013). It is predicted that future HSTs will be the major competitor against airliners in the sector of long-distance passenger transportation (Albalate et al., 2015).

Like other public transportation vehicles, HSTs are bringing convenience and efficiency to our daily lives, but also hastening regional and global spreads of communicable diseases (Gupta et al., 2011a). During past years, there have seen a large amount of travel-related global outbreaks of Tuberculosis (TB), Severe Acute Respiratory Syndrome (SARS) and Swine Influenza (H1N1) (Zhu et al., 2010). HST cabins play a crucial role in transmitting disease because they are closed indoor spaces with extremely high occupant density. Pathogen-carrying saliva droplets or nuclei released through cough or sneeze by an infected passenger have a higher possibility to be inhaled by other occupants compared with other indoor environments such as offices and dwelling houses. In order to minimize disease transmission and improve ride comfort, most modern HST cabins are ventilated using an up-to-down air-conditioning scheme, as shown in (Figure 4.1). This type of ventilation design sets inlets near the cabin ceiling and outlets near the floor, creating a descending overall airflow pattern which helps retain contaminants below passengers’ breathing zone. However, due to
the metabolic heat released by passengers, a strong upward buoyance flow is also created in cabins, which competes against the ventilating air flow and reduces the effectiveness of contaminant removal. In order to improve the effectiveness of contaminant control, a large flow rate of supplement fresh air is recommended (Zhuang et al., 2014), which however, would largely increase the energy consumption and degrade passengers’ thermal comfort. Therefore, optimized ventilation schemes, particularly carefully designed ventilation diffusers, have been developed to minimize disease transmission and improve the air quality in public transportation vehicle cabins while maintaining the ride comfort and reducing the energy consumption (Zhang et al., 2009a).

![Figure 4.1. A typical ventilation scheme in HST cabins](image)

As one of the crucial components of cabin ventilation systems, diffusers have been widely investigated both experimentally and simulatively. Daithankar et al. (2015) found the diffuser locations can influence passengers’ thermal comfort in a minibus cabin, and improper diffuser orientation angles would lead to a higher temperature for drivers and passengers next to windows. Zhu et al. (2012) simulated four types of
ventilation scenarios of a bus cabin including using linear diffusers and round diffusers. They found that diffuser type, filtration of recirculated air, location of exhaust vents and arrangements of passenger seats could all influence airborne diseases transmission. Chow (2002) studied the air-distribution performance in an enclosed train cabin. Their results indicated that the air diffusion terminal devices must be located at the upper level so that carbon-dioxide level in crowded train cabins can be well controlled at a low level. Zhang and Li (2012) found that in HST cabins, the back-door-oriented design of exhaust outlet coupled with the CRH3 model cabin inlet diffuser has strong ability to remove cough droplets. Wang et al. (2014) studied ventilation performance in three HST cabins including the CRH1, CRH2, and CRH5 models, which have similar cabin size and seat arrangements but different ventilation schemes. The results revealed that CRH5 has the best ventilation efficiency. Li et al. (2014) reported that on-the-market mainstream air diffusers in airliner cabins can be categorised as personal gaspers (on each passenger overhead), wall grilles (at the upper of the sidewalls) and roof grilles (at the top of aisle). These three types of diffuser have different ventilation characteristics regarding different cabins and working conditions. It is suggested that the mixed application of air diffusers would exert the best ventilation performance (Yan et al., 2015).

The aim of this study is to investigate the effects of diffuser types on the thermal flow field and contaminant transport in HST cabins. Computational fluid dynamics (CFD) models and numerical procedures for HST cabin simulations were first developed and validated using experimental data yielded from a mock-up cabin. Four types of prevailing HST cabin diffusers were subsequently assessed in terms of the thermal flow field and contaminant transport in a typical HST cabin model.

### 4.2 CFD simulation of HST cabin

HST cabin in this study was performed in the frame of the prevalent railway HVAC Standard TB1951-87 (MRPRC, 1987), EN13129-2016 (EU, 2016), UIC553 (UIC, 2005). In the simulations, the cabin was assumed to be fully occupied with 3D-scanned manikins. In order to minimize the computational cost while maintaining the accuracy,
simplified mesh-decimating-algorithm-based computer simulated persons (CSP) (Yan et al., 2016) were adopted in the HST cabin. Both gaseous and particulate contaminants considered in this study. Acetaldehyde, a typical human-released VOC in gaseous contaminant sector (Mochalski et al., 2014), was assumed to be released from manikin surface. For particulate contaminant, representing saliva drops were released by passenger cough.

4.2.1 Thermal Load Analysis

Based on the HVAC standards mentioned before, basic HST cabin HVAC operation requirements in summer seasons are shown in (Table 4.1). In order to obtain all the boundary conditions including supplement air temperature for ventilation simulation, it is necessary to calculate the thermal load in the HST cabin.

*Table 4.1. Summary of HST cabin HVAC operation requirements in summer (MRPRC, 1987, UIC, 2005)*

<table>
<thead>
<tr>
<th>Description</th>
<th>Requirement</th>
</tr>
</thead>
<tbody>
<tr>
<td>Mean Operation(ambient) temperature:</td>
<td>308K</td>
</tr>
<tr>
<td>Cabin interior(target) temperature setting:</td>
<td>300K</td>
</tr>
<tr>
<td>Permissible air speed in cabin:</td>
<td>0.08-0.6m/s</td>
</tr>
<tr>
<td>Sensible heat emitted by a human seated and wearing normal clothing:</td>
<td>63W</td>
</tr>
<tr>
<td>Temperature difference between inlet air flow and exterior:</td>
<td>&lt;8K</td>
</tr>
<tr>
<td>Temperature difference in same vertical line:</td>
<td>&lt;3K</td>
</tr>
<tr>
<td>Fresh air supplement rate:</td>
<td>20-25 m³/h/person</td>
</tr>
<tr>
<td>Fresh air ratio of total supplement air:</td>
<td>20-30%</td>
</tr>
<tr>
<td>Velocity of air flow at inlet and outlet vents:</td>
<td>&lt;3m/s</td>
</tr>
</tbody>
</table>
It is assumed that the thermal energy generated and absorbed inside a cabin is equal to the energy taken away by the airflow from inlet vents to outlet vents ($\dot{Q}_{\text{air}}$). The main sources of thermal load inside the cabin are the heat flux of human body ($\dot{Q}_{\text{human}}$) and thermal conduction of cabin wall ($\dot{Q}_{\text{wall}}$). Therefore, considering the energy conservation law, the thermal balance in the cabin can be expressed as follows:

$$\dot{Q}_{\text{air}} = \dot{Q}_{\text{wall}} + \dot{Q}_{\text{human}}$$  \hspace{1cm} 4.1

The thermal load of airflow can be calculated by

$$\dot{Q}_{\text{air}} = c \times \dot{m} \times (T_{t.i.} - T_{a.i.})$$  \hspace{1cm} 4.2

Where, $c$ is air specific heat capacity; $\dot{m}$ is the total air supplement rate; $T_{t.i.}$ is the target cabin air temperature, $T_{a.i.}$ is the supplement air temperature.

Considering the fact that heat transfer from cabin wall outer surface (CWOS) to its inner surface (CWIS) is equivalently a conduction process, the thermal conduction load of cabin wall can be expressed by

$$\dot{Q}_{\text{wall}} = \lambda \times s \times (T_{\text{CWOS}} - T_{\text{CWIS}})$$  \hspace{1cm} 4.3

Where $\lambda$ is the equivalent thermal conductivity of cabin wall, $\lambda = 2.22 \text{W/}(\text{m}^2\text{k})$ (Xu et al. 2011); $s$ is the area of the cabin wall; $T_{\text{CWOS}}$ is the temperature of CWOS; $T_{\text{CWIS}}$ is the temperature of CWIS. $T_{\text{CWOS}}$ is influenced by ambient temperature and solar radiation effects, thus,

$$T_{\text{CWOS}} = T_h + T_d = T_h + \frac{\rho J}{\alpha_W}$$  \hspace{1cm} 4.4

Where $T_h$ is ambient temperature; $T_d$ is equivalent temperature due to solar radiation effects; $\rho$ is the coefficient of CWOS absorbing radiation, $\rho = 0.7$ (MRPRC, 1987); $J$ is solar radiation intensity; $\alpha_W$ is the heat transfer coefficient of CWOS.
The sensible heat flux of human skin is the main heat loss from passenger bodies to the cabin (Melikov, 2015). Based on Standard TB 1951-87 (MRPRC, 1987), the convective heat flux of a person is calculated by

\[ \dot{Q}_{human} = \frac{n q_h}{A} \]  

Where \( n \) is the clustering coefficient of a fully occupied cabin and \( n=0.955 \) in this case. \( A \) is the surface area of a person exposure to air, \( q_h \) is the sensible heat emitted by a human seated with normal clothing. Considering the manikin skin intersects with seat, the effective manikin skin area for convective heat transfer is 1.01 m².

### 4.2.2 Mathematics Models

In this study, commercial software ANSYS CFX 16.0 was employed as the numerical solver to fulfil the models and methods. The cabin thermal field was solved using Navier–Stokes equations governing fluid flow and transport principles for incompressible turbulent flow in terms of mass, momentum and energy conservation equations, thus,

\[ \frac{\partial \phi}{\partial t} + \frac{\partial}{\partial x_i} \left( u_i \phi - \Gamma \frac{\partial \phi}{\partial x_i} \right) = S_\phi \]  

Where \( \phi \) is the general variable; \( u \) is the fluid velocity; \( \Gamma \) is diffusion coefficient; \( S_\phi \) is the source term. Equations for air turbulent were solved with RNG k-epsilon model and Boussinesq approximation was employed to address thermal buoyancy flow. The models have been maturely validated by many studies (Li et al., 2015a) and were thus not expatiated here.

Due to the extremely low concentration of gaseous contaminants, the transportation process was modelled using the drift-flux model:
\[
\frac{\partial \varphi}{\partial t} + \frac{\partial}{\partial x_i} (U \varphi) = \frac{\partial}{\partial x_i} \left( D\varphi \frac{\partial \varphi}{\partial x_i} \right) + S_{\varphi}
\]

4.7

Where \( U \) is the fluid velocity in air-flow domain; \( \varphi \) is the concentration of contaminant; \( S_{\varphi} \) is a volumetric source term; \( D\varphi \) is the kinematic diffusivity.

Particulate contaminants were tracked using the Lagrangian approach. For micron particles immersed in continuous air, important forces governing particle motions are the drag force \( F_D \) and buoyancy force \( F_B \) (Li et al., 2012b), thus,

\[
m_p \frac{dU_p}{dt} = F_D + F_B
\]

4.8

\[
F_D = \frac{1}{2} C_D \rho_F A_F |U_S| U_S
\]

4.9

\[
F_B = \frac{\pi}{6} d_p^3 (\rho_p - \rho_F) g
\]

4.10

Where \( m_p \) is the mass of particle, \( U_p \) is the particle velocity, \( C_D \) is the drag coefficient, \( \rho_F \) is the fluid density, \( A_F \) is the effective particle cross section, \( U_S \) is the velocity difference between fluid and particle, \( d_p \) is the particle diameter, \( g \) is the gravity vector.

For turbulent tracking, the instantaneous fluid velocity \( v_f \) is decomposed into mean \( v_f \), and fluctuating \( v'_f \) components. However, when drops hit the cabin wall, passengers or seats, the “Stick-to-Wall” model (ANSYS, 2016) was used in the simulation, which treats the walls as a sink term of the particulate phase.

### 4.2.3 Validation of the Mathematics Models

In order to validate the mathematical model and numerical procedures, the author has done the validation simulation based on a 7-row-seats mock-up cabin. The cabin was set up by Li et al. (2014), which can be used to mimic the densely occupied HTS cabin. Inside the cabin, seats were arranged by 3-3 with one aisle in the middle, and all the seats were fully occupied by manikins. Air was supplied from inlet diffusers at the top
of two side walls and exhausted from the outlets located at the bottom of side walls. The ventilation conditions were operated under ASHRAE standards (ASHRAE, 2004).

A corresponding computational model was set up based on the experimental conditions and simulated using above mathematic models. The velocity field of simulation results and experiment results in front of one row of manikins are compared in (Figure 4.2). The comparison of velocity magnitude and direction were similar between simulation results and experiment data near the diffusers area. However, there is a discrepancy in the area far away from the diffuser. This could be because of uncovered factors such as manikin individual difference. The comparisons of temperature and velocity between experiment and simulation on three representative locations inside cabin are shown in (Figure 4.3). It can be seen that the measured and simulated velocities agreed well with each other. The validation of temperature profiles show 1.5 °C deviations, and this value is 0.12 m/s for velocity profiles. Similar to (Figure 4.3), data from far-away sampling locations (location 2) had relatively larger deviations. Details of the validation are available at paper (Li et al., 2015b) and will not be repeated here. In general, the simulation results show good agreement with experimental results, and the mathematical model and numerical procedures have been proved to be acceptable for simulating the HST cabin ventilation environment.

![Figure 4.2. Comparison of velocity field between experiment and simulation in front of one-row passengers (Li et al., 2015b) ](image-url)
4.3 Effects of Diffuser Type

4.3.1 Diffuser Details

Four different types of diffusers were included in this study to investigate the effects of diffuser type on thermal flow and contaminant transport in HST cabins. These were the most common diffusers which are applied in Chinese CRH and Japanese Shinkansen HST. Details of diffusers’ geometrical structure and installation location are shown in (Figure 4.4).

- Diffuser Type 1: CRH2-oriented cabin diffuser. The inlet vents are on each cabin side wall below the luggage rack. In this simulation model, the inlet vent dimension is 720mm in length and 50mm in width. The space between each inlet vent is 480mm.

Figure 4.3. Comparison of velocity and temperature between experiment and simulation in 3 locations (Li et al., 2015b)
Diffuser Type 2: CRH3-oriented cabin diffuser. The inlet vents are on the roof of cabin, where a downward air supply can be executed. In this simulation model, the inlet vent dimension is 60mm in width and throughout the cabin. The space between each inlet vent is 980mm.

Diffuser Type 3: CRH5-oriented cabin diffuser. The inlet vents are on two sides of the roof, where air supply directions are against each other. In this simulation model, the inlet vent dimension is 60mm in width and throughout the cabin. The space between each inlet vent is 1100mm.

Diffuser Type 4: Shinkansen-oriented diffuser. Each inlet vents are located on each side of cabin wall, on top of every row seat and below luggage rack. It is based on a circular truncated cone sharp. In this simulation model, the supply air area of each inlet vent is 0.048m².

![Figure 4.4. Diffuser types and locations](image-url)
The airflow pattern in HST cabin is affected by a number of factors including air property, diffuser type, cabin geometry, seat arrangement, location of exhaust vents, arrangements of passengers and operation parameters (Zhu et al., 2012). In order to maintain the comparability, all the other conditions were assumed to be the same: same four-row second class cabin with seats arranged in 2+3 pattern based on CRH Model 2. In the simulation, acetaldehyde was chosen as representative VOCs, as it has a high concentration level compared with other components of VOCs emitted from human skin. The kinematic diffusivity of acetaldehyde is 1.6e-05 m2/s, and concentration measured from normal human skin is 3.0e-5 kg/m-3 VOC (Mochalski et al., 2014). In order to clearly observe gaseous contaminant dispersion, only passenger P2B was selected as acetaldehyde release person. For particulate contaminants, representative saliva drops were sprayed from passenger P2B’s mouth. In the initial situation, it was assumed that there were 2000 droplets with the drop radius of 3.5*10-6m and velocity of 3 m/s, which have been proven valid for representing human cough (Redrow et al., 2011). The basic dimensions of the simulation cabin and containments releasing source locations were labelled in (Figure 4.5).

![Figure 4.5. Simulation cabin and containments releasing source](image)

In this study, ANSYS ICEM 14.5 software was selected for meshing the HST cabin model. Due to the geometry complexity, unstructured tetrahedral girds were adopted in
all cases. For grid study, meshing independent testing had been done with several sets of simulation results based on different meshing approaches. Results indicated that 3.5 million mesh elements are sufficient for the issue of this study as a further increase up to 6.3 million mesh elements just caused a negligible velocity increase of 0.5%.

### 4.3.2 Ventilation Performance Analysis

When comparing the ventilation performance, two planes were chosen in the HST cabin. As shown in (Figure 4.5), the first plane is cross through the middle of the third-row passengers on the crosswise, while the second plane is cross through the middle of third column passengers on the lengthwise. These two plane locations can efficiently represent the ventilation characters in all types of HST cabins.

By comparing the velocity field simulation results (Figure 4.6) with HVAC standards (MRPRC, 1987, UIC, 2005), all of these four types of diffusers could fulfil ventilation requirements which states that permissible air speed in a cabin should be between 0.08m/s to 0.6m/s, and velocity of air flow at inlet and outlet vents should be less than 3m/s. The average temperature (Figure 4.6) inside the cabin is around 300k, temperature difference between inlet air flow and exterior is less than 8K, and temperature difference in any vertical line is less than 3k. However, all the four cases show the phenomenon that the air temperature around passengers is slightly higher due to the heat flux released from passengers. In addition, overall flow patterns are also affected by passengers due to the thermal plume. Therefore, eddy current airflow, which is harmful to exhausting contaminants, exits around some passengers.

For diffuser type 1, the velocity above baggage rack level is relatively high, while the best homogenous velocity field and temperature field stay below this level. Eddy current can be observed in upper cabin regions, but it barely affects sitting passengers. Also, the temperature gradient around passengers is relatively small. Hence body thermal plume observed in this case shows the least obviousness in all the four cases. For Diffuser type 2, even the fresh air comes downward from the inlet vents fitted on the cabin roof toward passengers sitting near the aisle, the air flow velocity around these passengers is not large. This is because of passengers’ heat flux effect that two
downward streams are combined into one in the aisle area and the velocity of the cool flow coming downwards is reduced. Type 2 diffuser exerts the problem that temperature distribution is not balanced between seat and aisle area, but this drawback will not significantly affect passenger comfort. As fresh air is directly supplied against each other near the cabin roof with diffuser type 3, a strong stream flows from the middle roof to middle aisle. Passengers in the third column are affected by this low-temperature and high-speed stream. Significant body thermal plume and eddy current around passengers near cabin wall sides are observed. It leads to a relatively high temperature around them. Overall velocity field in diffuser type 4 is small. Especially, a very weak velocity zone exists above baggage racks. It can hardly carry out the energy absorbed from cabin roof and therefore cause the temperature accumulation in this area. However, considering the fact that passengers maintain a sitting position during most of travelling time, the higher temperature zone in upper cabin area is not a critical problem. Because less air flows through the passengers sitting in the middle, average air temperature close to those passengers is the highest in the whole cabin. The lowest temperature is found around the passengers sitting near the cabin walls, as relatively strong fresh air blows down from diffusers located right above their heads.

Figure 4.6. Velocity and Temperature fields

In order to quantify the ventilation performance, Predicted Mean Vote (PMV), which is accepted as the most recognized index for thermal comfort (Gilani et al., 2015), was calculated based on the Fanger’s heat balance equation (ASHRAE, 2004). PMV in every passenger’s surface area was calculated from simulation results, and the average $\mu$ and variance $\sigma^2$ of PMV value of the whole HST cabin section were calculated by,
$$\mu = \frac{\sum X}{N}$$  \hspace{1cm} 4.11

$$\sigma^2 = \frac{\sum (X - \mu)^2}{N}$$  \hspace{1cm} 4.12

where, X represents the individual PMV value of each passenger and N is the passenger number. The results are plotted in (Table 4.2). As it is impossible to perfectly satisfy people’s requirements on thermal environment due to diverse individual thermal conception, a criterion is set up by ISO 7730 (ISO, 2005) that there are different classes of satisfaction: 1) Class A: range of PMV(-0.2 < PMV < +0.2) is the highest satisfaction of environment; 2) Class B (-0.5 < PMV < +0.5) is the moderate requirement of satisfaction level; 3) class C (-0.7 < PMV < +0.7) is the minimum requirement of examination criterion within the thermal comfort criterion. All four types of diffuser applied in CRH2 cabins could highly meet the thermal comfort criterion. Particularly, diffuser type 3 meets the Class A standard. Besides, diffuser type 3 has the lowest variance value of PMV, which means each passenger could perceive the most similar thermal comfort. Diffuser type 4 shows a relatively large variance value, which indicated that seats at different positions sustain large thermal differences.

<table>
<thead>
<tr>
<th>PMV</th>
<th>Diffuser type 1</th>
<th>Diffuser type 2</th>
<th>Diffuser type 3</th>
<th>Diffuser type 4</th>
</tr>
</thead>
<tbody>
<tr>
<td>Average</td>
<td>-2.48E-01</td>
<td>-2.30E-01</td>
<td>-5.01E-02</td>
<td>-2.79E-01</td>
</tr>
<tr>
<td>Variance</td>
<td>3.45E-02</td>
<td>4.79E-02</td>
<td>1.57E-02</td>
<td>1.45E-01</td>
</tr>
</tbody>
</table>

4.3.3 Gaseous Contaminants Dispersion Analysis

The distribution of VOCs released by passenger P2B was studied to evaluate the capabilities of different diffusers on removing occupant-generated pollutants. In this study, the VOC concentration was normalized in terms of the release concentration at the manikin P2B skin. VOCs concentration rendering (Figure 4.7) indicates the overall
distribution performance of steady-state VOCs ventilated by four types of diffusers. For four type diffuser applied cabins, VOCs mainly concentrate around the source passenger P2B, above the source passenger P2B, above-behind the source passenger P2B, and around neighbourhood passengers P2B respectively.

In order to clearly analysis diffusers’ ability to control the spread of VOCs, the isosurface areas of VOCs concentration at 0.05 are plotted in (Figure 4.8). It is clearly illustrated that Diffuser type 1 has the best ability for preventing the VOCs from spreading to other passengers as the isosurface volume is the smallest. VOCs emitted from passenger P2B can be efficiently carried out by air flow rather than spreading to neighbour passengers. Type 2 diffuser ventilation system has very similar performance compared with Type 1. However, some VOCs are concentrated in the area above passenger P2B. In Type 3 diffuser simulation, little VOCs are spread to passengers sitting close to windows, but the area above baggage rack gathers relatively high concentration VOCs. Fortunately, this is not a passenger-active area. The isosurface area of VOCs in Diffuser type 4 is the largest, that both neighbour passengers P2A and P2C are exposed in it.

According to (Figure 4.7 and Figure 4.8), it can be concluded that the transport and distribution characteristics of released VOCs are highly sensitive to the diffuser type under the same other conditions. Therefore, the health risks impacted by gaseous contaminants released from a sick passenger may vary depending on the applications of different ventilation systems. In order to achieve a quantitative exposure risk assessment, VOCs concentration in each manikin’s breathing zone was analysed. It is defined in the Australia Work Safety Standard (SafeWork, 2011) that the personal breathing zone is a hemisphere of 300-mm radius extending in front of the face and it is measured from the midpoint of an imaginary line joining the ears. Similar to PMV, the average and variance values of VOCs concentration in each passenger’s breathing zone are illustrated in (Table 4.3). It is clear that the average VOCs concentration is not as sensitive to the diffuser as VOCs distribution process. According to (Table 4.3), Diffuser type 1, 2, and 3 have similar average VOCs concentration value, while Diffuser type 4 has a relatively larger average value. Besides, the variance value
indicates Diffuser type 1 and 4 have a relatively unequal VOCs concentration in each passenger breathing zone. Especially for Diffuser type 4, the large average and variance values suggest some passengers are exposed in high gaseous contaminants concentration environment. This phenomenon can be validated in the VOCs concentration rendering figures (Figure 4.7).

*Figure 4.7 Normalised VOCs concentration*
Table 4.3. VOCs concentration in breathing zone of four type diffusers used cabin

<table>
<thead>
<tr>
<th>VOCs concentration</th>
<th>Diffuser type 1</th>
<th>Diffuser type 2</th>
<th>Diffuser type 3</th>
<th>Diffuser type 4</th>
</tr>
</thead>
<tbody>
<tr>
<td>Average</td>
<td>5.28E-03</td>
<td>5.16E-03</td>
<td>5.54E-03</td>
<td>7.78E-03</td>
</tr>
<tr>
<td>Variance</td>
<td>6.52E-05</td>
<td>1.77E-05</td>
<td>1.42E-05</td>
<td>9.29E-05</td>
</tr>
</tbody>
</table>

Figure 4.8 VOCs concentration at 0.05 isosurface
4.3.4 Particle Contaminants Dispersion Analysis

The transport and distribution characteristics of particles generated by coughs from passenger P2B were also studied with different diffusers. Separate computations were conducted to simulate the tracks of particles, as illustrated in (Figure 4.9). It is obvious that the diffuser type differences would not only significantly influence the spread ability of gaseous containments, but also dramatically changes the trajectories of particles transport. (Figure 4.9) demonstrates that particle trajectories are jointly controlled by the ventilating airflow and the buoyancy-driven thermal plume. The particles, especially in the regions near the manikin bodies, would be elevated to a higher level, descend towards floor, or even be highly concentrated around passengers, depending on the intensity of the thermal plume.

For Diffuser type 1, the particles were predicted to be quickly dispersed from passenger P2B, joined the overall flow vortex and eventually travelled widely in the cabin. During this process, many droplets were stuck on the window beside passenger P1B and the roof area above the source passenger. The model predicted that in the cabin configured with Diffuser type 2, the particles exhaled by passenger P2B would mainly concentrate in the bulk air above the passenger, thus passengers around P2B were theoretically located in a clean region. Then most droplets were travelled through the area around passengers P3C to the outlet vent under passenger P3B. After leaving the release point in Diffuser type 3 cabin, the particles quickly concentrated into the bulk air flow to the back area and distributed into a wide range. It is clear that a high-concentration zone exits in front of passenger P3D and P3E due to eddy flow. Most droplets were ended by attaching the cabin wall. For Diffuser type 4, the exhaled particles travelled with very limited distance rather than spread widely through the entire cabin section. The droplets are Droplets were largely ended around passenger P2A, P2B and P2C, with the rest mostly taken away by the outlet vent under passenger P2B.

(Figure 4.9) also illustrate the transient characteristics of particle transportation in the cabin environment under different diffuser conditions. The transient particle distributions from \( t = 1 \) to 60s are shown based on particles colours. It demonstrates
that dispersion speed and range of particles through the air are highly sensitive to
diffuser type differences. When the particles were released in Diffuser type 1, they
moved towards the right side with the airflow, then suddenly changed their direction
backwards when they reached the large flow (t = 3 s). At t = 20s, the particles were
fully dispersed to the whole cabin space. Comparatively, the dispersion speed of the
particles released from Diffuser type 2 is much higher. At t = 3s, the particles reached
the upward bulk region, while at t = 20 s, some particles were concentrated around
Passenger P3C. Similar with Diffuser type 2, particles concentrated into an upward
bulk in the beginning stage of particle transportation in Diffuser type 3. But at t = 20 s,
particles fully dispersed into the back area of Passenger P2B. The figure of Diffuser
type 4 from t = 3s to 20s illustrated a long-time concentration phenomenon of particles
around passenger P2A, P2B and P2C. However, it can be easily found that after 60
seconds, all these four types of diffusers applied cabins had less than 10% residual
droplets in the air. The simulation results revealed all these four types of diffusers could
efficiently remove the particle contaminants within a certain time period, while particle
distribution processes vary.

Under the condition of four types of diffusers, the average and variance value of
particle averaged volume fractions in the breathing zone were calculated for
quantifying the particle distribution performance. According to (Table 4.4), Diffuser
type 2 and type 3 have relatively small average and variance values. It indicates a
reasonable particle distribution performance in breathing zone. Comparing (Table 4.4)
with (Table 4.3), average and variance values of particle distribution in passengers’
breathing zone were noticeably different from VOCs distribution performance. Diffuser
type 4 had relatively small average and variance values of particles in breathing zone,
while the average and variance value of VOCs concentration values were the largest of
all the four types of diffusers. Diffuser type 1, which had good performance in VOCs
distribution, performed unideal in droplets distribution.
Figure 4.9 Particle transport and distribution at various time steps

Overall, all these four type diffusers could meet the industrial standards of HST ventilation. This study also provides information which is not covered by standards, including thermal comfort difference, gaseous contaminants distribution, particle contaminants transport and breath zone air quality analysis. In general, design of different type of diffusers should be a crucial consideration within any HST cabin design.
Table 4.4. Particles averaged volume fraction in breathing zone of four type diffusers used cabin

<table>
<thead>
<tr>
<th></th>
<th>Diffuser type 1</th>
<th>Diffuser type 2</th>
<th>Diffuser type 3</th>
<th>Diffuser type 4</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Average</strong></td>
<td>6.43E-03</td>
<td>1.48E-03</td>
<td>9.94E-04</td>
<td>3.67E-03</td>
</tr>
<tr>
<td><strong>Variance</strong></td>
<td>6.36E-04</td>
<td>1.29E-05</td>
<td>4.55E-06</td>
<td>1.13E-04</td>
</tr>
</tbody>
</table>

### 4.4 Conclusions

This study employed a section model of CRH2 cabin containing four rows of seats with fully occupied passengers to investigate the effects of diffuser type on airflow field and contaminant transport characteristics. Simulations were conducted with consideration of cabin thermal load analysis and mathematic models were validated using experimental data from a mock-up cabin. Conclusions arose from this study are as follows.

- The geometry and location of diffusers have significant effects on ventilation performance and contaminant transportations in HST cabins. The overall and local airflow fields vary, while all PMV indexes are ideal. Similarly, contaminants distribution process varies, all four type diffusers have a low concentration of gaseous contaminants and can efficiently remove the particle contaminants in a certain time. If only considering breathing zone area, Diffuser type 3 has the best containment distribution performance, because the average and variance values of VOCs concentration and droplets volume fraction are the smallest among four types of diffusers.

- Gaseous contaminants and particle contaminants transport performances are different under the same ventilation condition. For instance, Diffuser type 1 has better VOCs distribution performance in passenger breathing zone compared with Diffuser type 4, but the particle distribution performance is worse.

- This study provides a more comprehensive way to assist assessing the diffuser efforts in ventilation schemes, in terms of not only thermal flow fields, but also
contaminants removal, which could be a supplement to the current industrial standards.
Chapter 5

The Effects of Cough-jet on Airflow and Contaminant Transport in a Cabin Section

The main findings of this chapter have been included in:


The goals of this study were to investigate the effect of cough-jet on local airflow and contaminant transport in a typical cabin environment by using CFD. A fully occupy cabin section was employed as the computational domain. Contaminants were released through coughing passengers from different locations inside the cabin. Numerical results in terms of contaminant transport characteristics were examined and compared. It can be concluded that cough-jet has significant effects on air flow in front of cough-passenger in short period of time. Also, it was found that, without considering the cough-jet model, the simulation results could not be a precise representation of the transport and distribution of cough-generated airborne contaminants.
5.1 Introduction

As more than two billion people are traveling in commercial flights each year (Gendreau and DeJohn, 2002), great concern on the inflight infection disease transmission has been raised. During the global outbreaks of Tuberculosis (TB), Severe Acute Respiratory Syndrome (SARS) and Swine Influenza (H1N1), the risks of airborne infection spread among the passengers inside aircraft cabins were found to be very high (Mangi and Gendreau, 2005). In order to effectively minimise the disease desperation in the commercial cabin and improve the ventilation performance, Computational Fluid Dynamics (CFD) was utilised as a cost-efficient tool to investigate and identify various affecting factors, such as ventilation scheme, the manikin thermal plume and human activities (Zhang et al., 2005, Spitzer et al., 2010, Salmanzadeh et al., 2012).

When passengers were coughing, most attentions were drawn to the aforementioned affecting factors, whilst one of the factors that released simultaneously with the contaminants through the coughing process, the cough-jet, was overlooked. Most existing studies assumed the cough-jet were ignorable (Chen et al., 2013), as the cough-jet could be quickly damped after release. For instance, Li et al. (2016a) numerically investigated the airborne contaminant distribution realised from one manikin’s mouth in a 7-row airliner cabin section. In their study, only contaminants were released though coughing without generating any cough-jet. This was also because it would require a significantly high computational cost to investigate the cough-jet characteristics in a full-scale airliner cabin due to the extreme complexity of the cabin environment. According to Gupta et al. (2011b), tracking coughing droplets transport for 4 minutes in a 7-row cabin took more than 4 weeks to finish using a fully operated computer cluster.

On the other hand, the cough-jet even in the first few seconds was argued as an important factor on the local airflow distribution from other indoor spaces. Kwon et al. (2012) pointed out that accurate description of the initial velocity distribution from coughing process is particularly important in the study of the fluid dynamics property of the respiratory particles. Their study concluded that only considering the airflow...
field without the cough-jet may not be an appropriate representation of the transport and distribution of airborne contaminants. By experimental studied the concentration of contaminants in aircraft cabin during several long-distance flights, Dechow et al. (1997) illustrated that the peak concentrations of bacteria can be significantly observed after coughing from the passenger. Gupta et al. (2011a) also argued that the cough-jet driven by the coughing process should be considered as one of the most important factors in airborne disease transmission. In public transport cabins, due to the densely occupied environment, cough-jet released by source passengers could break the local airflow not only in their but also their neighbours’ breathing zone. Thus, it is crucial to include the cough-jet when investigating the contaminants transport and distributions through coughing in the cabin environment. However, the overlook of the cough-jet in the cabin environment was not yet drawn enough attention in the existing literature.

Therefore, in this study, the effect of cough-jet on local airflow and contaminant transport was thoroughly investigated in a typical median-size cabin environment by using CFD. A cabin section based on Boeing 737, which initially used in our previous research (Yan et al., 2015), was employed in this study to continuously investigate the cough-jet effect on top of the other important studied factors such as thermal plume. Contaminants were released through coughing with various size distributions in conjunction with the simultaneous production of cough-jet to imitate the real coughing process. As the difference in the source locations may lead to a significant difference of contaminant fields in airliner cabin (Li et al., 2014), cough-jet released from different coughing passengers were quantitatively analysed. The numerical results were also compared to our research outcomes under the same cabin environment to identify the importance of the local cough-jet.

5.2 Methods and Validation

5.2.1 Computational Domain and Boundary Conditions

Considering that the higher occupant density in an economy-class cabin may cause a higher risk of disease transmission, this study focuses on the particulate containments transport and distribution in economy-class cabins. The economy cabin model was built
based on a typical medium-size public transport cabin (Boeing 737). A cabin section with three seats and three Passenger A, B and C were built as the computational domain, as illustrated in Figure 5.1. According to Liu et al. (2012b) experimental measurement, the airflow pattern is approximately symmetric across the central plane along the aisle. Therefore, a symmetric boundary was set up in the central plane along the aisle to save the computational cost. For the same purpose, two translational periodicity boundaries were applied at the front and back planes of the cabin section, since the ventilation layout and geometry arrangement were uniform at each row. By applying symmetric and periodic boundaries, the computational domain was largely trimmed down which could significantly reduce computational cost, while more computational resource was spent on investigating the local cough-jet. Also, as the cough-jet would merge with the surrounding airflow within a few seconds after release (Chen et al., 2013), the reduced cabin section was tested to be sufficient enough to capture the cough-jet behaviour.

Figure 5.1 Simulation cabin section and the grid size around the mouth area.

The ventilation schema was built based on Liu et al.’s experimental cabin mock-up and the ventilation rate was set according to the ASHRAE aviation standard (Liu et al.,
Air was supplied from the inlet diffusers at the upper side of the cabin walls and exhausted from the outlets located at lower side of the cabin walls. The air supply rate through the inlets was 0.04 kg/s according to the ASHRAE aviation standard with an inlet air temperature of 25 °C (ASHRAE, 2009). Three detailed 3D-scanned adult female manikins model with seating posture were applied as passengers in the cabin section (Yan et al., 2016). The mouth area of each manikin was further modified in Figure 5.1, in which the cough exhaust boundary was set in the simulation. An equivalent convective heat load of 40W was set on each manikin to imitate the passengers’ metallic body heat, which matches the settings from existing literature and our previous study (Topp et al., 2002, Yan et al., 2015). All other solid walls, including cabin ceiling, side walls, floor and seats were assumed to be the adiabatic condition.

5.2.2 CFD model

In this study, the cabin airflow field was solved using the incompressible Navier-Stokes equations. Zhang et al. found that the renormalization group (RNG) k-epsilon model can effectively predict the turbulent feature of the airflow in the cabins (Zhang et al., 2009b, Yakhot and Orszag, 1986). Therefore, air turbulent was based on RNG k-epsilon model. Boussinesq approximation was employed to address thermal buoyancy flow induced by human body heat. All governing equations were solved using the commercial CFD software ANSYS CFX 16.0.

For cough-jet model, particulate contaminants were assumed to be exhaled from the passengers’ mouth area through coughing. During one cough, cough-jet was released simultaneously with the contaminants. An average cough-jet rate was plotted in Figure 5.2, based on tests (Gupta et al., 2009). Various sizes of contaminants were considered in this study, based on the size distribution of contaminants from (Bourouiba et al., 2014). The profile of droplet size distribution was given in Table 5.1, in which the exhaled contaminants sizes are mainly concentrated from 5μm to 25μm.
Figure 5.2 Cough-jet flowrate of a single cough based on experimental testing (Gupta et al., 2009)

Table 5.1 Discrete probability of particle size during one coughing.

<table>
<thead>
<tr>
<th>Diameter(μm)</th>
<th>2</th>
<th>4</th>
<th>8</th>
<th>16</th>
<th>24</th>
<th>32</th>
<th>40</th>
<th>50</th>
</tr>
</thead>
<tbody>
<tr>
<td>Number</td>
<td>44</td>
<td>284</td>
<td>966</td>
<td>1592</td>
<td>865</td>
<td>415</td>
<td>235</td>
<td>105</td>
</tr>
<tr>
<td>Diameter(μm)</td>
<td>75</td>
<td>100</td>
<td>125</td>
<td>150</td>
<td>200</td>
<td>250</td>
<td>500</td>
<td>1000</td>
</tr>
<tr>
<td>Number</td>
<td>136</td>
<td>79</td>
<td>43</td>
<td>33</td>
<td>31</td>
<td>27</td>
<td>29</td>
<td>13</td>
</tr>
</tbody>
</table>

Based on the research of Zhang and Chen (2007), for the unsteady scenario, the prediction of airborne particle distribution has a better tracking accuracy by using Lagrangian approach. For micron particles immersed in continuous air, important forces governing particle motions are the drag force $F_D$ and buoyancy force $F_B$ (Li et al., 2012b), thus,
\[ m_p \frac{dU_p}{dt} = F_D + F_B \]  

5.1

\[ F_D = \frac{1}{2} C_D \rho_F A_F |U_S| U_S \]  

5.2

\[ F_B = \frac{\pi}{6} d_p^3 (\rho_p - \rho_F) g \]  

5.3

Where \( m_p \) is the mass of particle, \( U_p \) is the particle velocity, \( C_D \) is the drag coefficient, \( \rho_F \) is the fluid density, \( A_F \) is the effective particle cross section, \( U_S \) is the velocity difference between fluid and particle, \( d_p \) is the particle diameter, \( g \) is the gravity vector.

For turbulent tracking, the instantaneous fluid velocity \( U_f \) is decomposed into mean velocity \( \bar{U}_f \), and fluctuating eddy velocity \( U'_f \) components, thus,

\[ U_f = \bar{U}_f + U'_f \]  

5.4

Considering that most surface area of cabin walls, seats, and passengers’ cloths are made of fabrics, droplets could not bounce back to the fluid domain after collisions. Therefore, the “Stick-to-Wall” model (ANSYS, 2016) was used in simulation for this phenomenon, which treats these surfaces as a sink term of the particulate phase.

Due to the fact that cough-jet ejected from manikin mouth is an unsteady process, the transient simulation was conducted in this study. According to Figure 5.2, cough begins with a very high acceleration exhalation flow rate for a very short time. When at peak velocity time (PVT) 0.08 second, the cough peak flow rate (CPFE) reach at 4.2 L/s. Then, exhalation flow rate decreases to the zero in 0.53 seconds, where the cough expiratory volume (CEV) is 0.87 L. Therefore, in the large gradient stage from 0s to 0.15s, time was dived into small computational time-steps that high accuracy of droplet ejection, along with cough-jet, can be captured.

For the simulation domain, ANSYS ICEM was used to generate the unstructured mesh. As the occupied manikins have detailed features, high-resolution meshes were built inside the simulation domain. Also, inflation layers were applied in all surfaces to
capture the accurate flow movement, as illustrated in Figure 5.1. Grid independency test was conducted over coarse meshing to dense meshing. The independency test show that the air flow velocity becomes stable, when the mesh elements number is over 1.9 million in this simulation domain.

Liu et al.’s experiment data in the first-class cabin of an MD-82 cabin was used to validate the current numerical model (Liu et al., 2012b). In their experiment, measurements of airflow velocity were conducted by using particle image velocimetry (PIV) in a vertical plane located in front of manikins. The predicted airflow field in Plane X-Z was compared with the PIV experimental results in Figure 5.3. Figure 5.3 demonstrated that the overall airflow patterns yielded from the adopted experiment and our numerical predictions were very close. Both experimental and numerical results show large magnitude velocities existing within the middle region. However, the direction differences of these velocities could be observed. This could be attributed to the geometry difference between numerical model and test cabin. More details of numerical model validation can be referred in previous study (Yan et al., 2015). In general, the simulation results show good agreement with experimental results, and the mathematical model and computational model have been proved to be acceptable for simulating the cabin ventilation environment.

Figure 5.3 Comparison of airflow vectors between experimental data and computational results (Yan et al., 2015).
5.3 Results and Discussion

5.3.1 Cough-jet Effects on Fluid Flow

In this study, three computational cases (Case-1, Case-2 and Case-3) were conducted in regard to coughing from Passenger-A, Passenger-B, and Passenger-C respectively. In order to explore the cough-jet effects on cabin airflow, the local airflow fields around passenger B’s breathing zones of Case-2 were firstly investigated (SafeWork, 2011). The velocity in the vertical plane and horizontal plane crossing passenger B’s breathing was illustrated in Figure 5.4. In steady state, the breathing-zone was controlled by ventilation jet, human thermal plume and manikin geometry (Yan et al., 2015). However, when cough-jet was excluded from the models, the breathing-zone was completely controlled by the cough-jet. Two specific time-step states were plotted in Figure 5.4. At 0.08s, which is the PVT in Figure 5.2, the velocity magnitude of airflow in breathing zone was much larger than in the steady state. The maximum velocity magnitude reached 11.2m/s, which is almost 50 times of steady air-flow velocity. It also can be identified clearly that a large vortex generated by cough-jet in breathing-zone around this time. At 0.5s, when cough-jet ends, the velocity magnitudes of airflow in breathing zone also have the trend of reducing to the similar magnitude of steady state. Therefore, the duration and development of cough-jet have significant effects on local flow around breathing-zone.

For a more comprehensive study of cough flow performance on different airflow fields, all three cases in regard of coughing from three different locations were analysed together. The vector figures of airflow velocities in the plane crossing the coughing passenger body were presented and compared at different time steps in Figure 5.5.

When the cabin ventilation flow was steady (Figure 5.5, T=0s), the overall airflow pattern was dominated by the inlet air coming from the diffuser located on the upper side wall. Uprising buoyancy flows can be observed above all the passenger heads due to thermal plume effects. Local airflow filed in every passenger’s breathing-zone was slightly different. An uprising airflow was monitored in front of Passenger-A who sat close to the wall, while a downward airflow was observed in the breathing zone of
Passenger-C who was next to aisle. Between them, a small vortex flow existed around the Passenger-B.

Figure 5.4 Velocity vector field around breathing-zone for cough passenger B in Case-2.

Although all the three passengers suffer different airflow patterns in their breathing zones at steady state, similar cough-jet patterns can be clearly identified during the first 0.5s of cough. From Figure 5.5, T=0.08s, Case-1 and Case-3 have almost the same flow pattern of Case-2 discussed in Figure 5.4, that large vortex is observed. Besides, all these three planes indicated that direction of cough flow would not be affected by the ventilation airflow during this period as it was perpendicular to the back of front seat. At the time of 0.5s, when the cough-jet was ended from generated by passenger, all the airflow velocity of three cases decreased largely in breathing-zone. Therefore, during
the first 0.5s, no matter of the difference between local airflow fields, the strong cough-jet dominated the airflow. Within this time period, the affected distance reached 0.45m from coughing mouth. However, in the region where cough-jet did not reach, such as upper cabin or passenger leg area, no infection was observed.

After 0.5s, cough-jet had less effect on cabin airflow due to even longer distance it had travelled, while the cough-jet pattern varied of each case. Case-1 cough-jet joints the uprising ventilation flow caused by the thermal plume. The joint flow would then mix with flash airflow supplying from inlet diffusers and it would distribute into the whole cabin. In Case-3, an opposite cough-jet direction downward to the bottom of front seat can be seen. In Case-2, half of the cough-jet joined the uprising ventilation airflow while the other half moved down. After 2s, the cabin airflow pattern became reasonably steady in all three cases. Therefore, during the period of 0.5-2s, the affected airflow region reached the front seat which was 0.65m away from coughing mouth. Within this region, cough-jet did not dominate the airflow anymore. Instead, the domination of airflow would be gradual changed by the cabin ventilation airflow.

<table>
<thead>
<tr>
<th>Case</th>
<th>T=0(steady)</th>
<th>T=0.08s</th>
<th>T=0.5s</th>
<th>T=1.5s</th>
<th>T=2s</th>
</tr>
</thead>
<tbody>
<tr>
<td>Case-1</td>
<td><img src="image1" alt="Image" /></td>
<td><img src="image2" alt="Image" /></td>
<td><img src="image3" alt="Image" /></td>
<td><img src="image4" alt="Image" /></td>
<td><img src="image5" alt="Image" /></td>
</tr>
<tr>
<td>Case-2</td>
<td><img src="image6" alt="Image" /></td>
<td><img src="image7" alt="Image" /></td>
<td><img src="image8" alt="Image" /></td>
<td><img src="image9" alt="Image" /></td>
<td><img src="image10" alt="Image" /></td>
</tr>
<tr>
<td>Case-3</td>
<td><img src="image11" alt="Image" /></td>
<td><img src="image12" alt="Image" /></td>
<td><img src="image13" alt="Image" /></td>
<td><img src="image14" alt="Image" /></td>
<td><img src="image15" alt="Image" /></td>
</tr>
</tbody>
</table>

*Figure 5.5 Velocity vector field across X-plane for different cough passenger.*
5.3.2 Cough-jet Effects on Droplets Distribution

The transport and distribution characteristics of particles generated by the coughs from the passengers were studies under both with cough-jet and non-flow cough conditions. Separate computations were conducted to simulate the tracks of the particles with size of 4μm exhaled by each of the three passengers. According to Figure 5.5, the velocity magnitude of cough-jet was much larger than cabin ventilation airflow. As the particles were carried by airflow, the distribution process in non-flow cough condition was much slower than in cough-jet condition. Therefore, in this study, the particle tracking times were observed for 40 seconds in non-flow cough condition and 10 seconds in cough-jet condition, as illustrated in Figure 5.6. It is clear that the passenger cough-jet not only had a significant effect on the airflow field as illustrated in Figure 5.5, it also dramatically altered the trajectories of particle transport. Figure 5.6 also demonstrated that the particle trajectories were jointly controlled by the ventilating airflow and cough-jet. The particles would be elevated to a higher level, descend toward the floor, or sunk in the surface of seats, depending on the location of coughing passenger.

When the particles were released by Passenger-A (Figure 5.6.a) without cough-jet, they immediately move upward with the airflow, and then suddenly changed their way downward when they reached the inlet diffuser as the particles were carried on by the fresh airflow. The particles were travelling with limited distance rather than travelling widely through the entire cabin section. Gradually, the particles formed a lock-up circle in front of Passenger-A and Passenger-B. However, when cough-jet was included in the model, the particles distribution trajectories were significantly different. Figure 5.6.b illustrated that the released particles were mainly carried by cough-jet until they reached the front seat. With part of particles sunk into the seat surface, the rest were carried by the arsing airflow to the inlet diffuser area. Then the strong fresh airflow brought these particles into the aisle area. Due to the balance of the uprising thermal plume from front passengers against the descending ventilation airflow, the particles lock-up as shown in Figure 5.6.a was not existed in this situation.
Figure 5.6.c and Figure 5.6.d illustrated the trajectories of particles exhaled by Passenger-B, who was sitting in the middle of the seats. After released by non-flow cough from Passenger-B, the particles quickly dispersed and joined the overall flow vortex and fully dispersed in the whole cabin section. However, when cough-jet was included in the model, most particles were predicted to be carried by strong cough-jet, ended by sunk into the front seat (Figure 5.6.d). Only very little particle carried by uprising airflow can be observed. Therefore, both Passenger-A and Passenger-C was located in a region without particles.

The transport trajectories of particles exhaled by Passenger-C were illustrated in Figure 5.6.e and Figure 5.6.f. It was noticed that when cough-jet was excluded, the model predicted that the particles exhaled by Passenger-C were mainly concentrated in the bulk air above the Passenger-C (Figure 5.6.e). By considering cough-jet, the particles were predicted to be distributed in a different range (Figure 5.6.f). Half of particles were sunk into the front seat the same as Case-1 and Case-2. The other particles fell into the leg and seat area of coughing-passenger following parabola trajectories.

![Figure 5.6 Particle trajectories for without cough-jet and with cough-jet cases.](image)
In order to quantitative analysis the cough-jet influence on particles, comparisons of average particle traveling distance in 13s was plotted in Figure 5.7. From Figure 5.7, the dispersion of particles was quite sensitive to the release location. For instance, particles have the longest travel distance when they were released from Passenger-A in both with cough-jet and without cough-jet scenarios. However, in all three cases, particles could travel further and faster in a short period of time when cough-jet was taken into simulation. Especially in Case-3, the average traveling distance in with cough-jet situation is 3 times longer than the without cough-jet situation. Therefore, the large difference of particle travelling distance has a significant influence on the air-borne disease transmission.

![Graph showing particle travelling distance in 13s for without cough-jet and with cough-jet cases.](image)

Figure 5.7 Particle traveling distance in 13s for without cough-jet and with cough-jet cases.

An overall review of Figure 5.6 and Figure 5.7 demonstrated that as cough-jet was included in the model, particles were mainly carried by cough-jet for a certain time. The strong cough-jet caused significant changes in the particle trajectories from those yielded from the non-flow coughing cases. According to Figure 5.5, before 0.5 second, cough-jet dominate the particle transportation, while after 1.5 seconds, cabin ventilation airflow dominates the particles distribution. Also, it demonstrated that dispersion speed and range of particles through the air were highly sensitive to their release locations, which affected by the local ventilation airflow pattern.
5.3.3 Different Droplet Sizes Distribution Analysis

According to Table 5.1, the quantity fraction of particle in different diameter was distributed in Figure 5.8. According to previous research (Mangili and Gendreau, 2005), the disease transmission rate could depend on saliva mass. Therefore, the mass fraction of droplets in various sizes was also plotted in Figure 5.8. By comparing the quantity fraction and mass fraction of different particle size, it is clear that in one cough process, even 95% particles had the size smaller than 100 μm, the total mass value of these particles only accounts for 2% of entire particle mass. While the large particles have a very small amount, the mass fraction was significant. So, from this point of view, it is necessary to discuss the transport and distribution characteristics of different size particles. Based on these, the different-size particles exhaled during cough from three passengers were studied. Simulation results were plotted in Figure 5.9. The trajectories of different particle sizes were also marked in this figure using different colours.

![Figure 5.8 Particle number and mass distributions of various sizes.](image)

When comparing the distribution of particles with diameter larger than 5E-5m, it can be seen that trajectories of these particles are similar in all three cases. Particles sized between 5E-5m and 2E-4m fell into the leg and seat area of coughing-passenger following parabola trajectories, immediately after leaving the mouth. The particles with
size larger than 2E-4m have similar distribution trajectories as particles with size between 5E-5m and 2E-4m. However, these particles fell further to the shoes area or sunk into the back of front seat. Therefore, a conclusion can be made that cough-jet and cough location have small effect on the trajectory of particle transport if the diameter of particle is larger than 5E-5m.

However, for particles with diameter smaller than 5E-5m, the cough-jet and cough location would significantly affect the trajectories of particles. In Case-1, these particles moved forwards along with the cough-jet after released by the Passenger-A, and then changed their direction upwards when they reached the front seat. Case-2 model predicted that particles with sized smaller than 5E-5m would mainly join the cough-jet ended by stucked on the front seat. Similar to Case-2, these particles were carried by the cough-jet to the lower region of front seat. Overall, the trajectories of particles in this size range were similar to the flow pattern as shown in Figure 5.9.

![Figure 5.9 Particle transport and distribution of various sizes.](image)

The main reason small droplets and large droplets travelled through significantly different trajectories can be explained by Lagrangian-method. In Lagrangian approach, small droplets were dominated by the drug force caused by airflow, while large droplets were dominated by the gravity caused by self-weight. Based on above discussion, the droplets with size larger than 5E-5m were mainly deposited in the leg
area of coughing passenger. Therefore, even these large droplets may have higher probability of spread disease, they were mainly descended in a short time and could not be transported into other passengers breathing-zone. It can be concluded that in the simulation of cough, it may only need considering the droplets with size smaller than 5E-5m.

Figure 5.10 Average particle residence time in the passengers’ breathing zones.

According to Figure 5.6, under a given airflow field, the transport and distribution characteristics of exhaled particles were highly sensitive to the location of release. Therefore, the health risk impacted by the particles released by a sick passenger on other passengers may be different. In order to achieve a quantitative result of exposure risk assessment, the particle residence time in each manikin’s breathing zone was analyzed. The averaged residence time of the particles exhaled by a passenger in other passengers’ breathing zone was illustrated in Figure 5.10. It was clear that the particle residence time is sensitive to both the location of release and the interested breathing zone. According to Figure 5.10, the particles exhaled by Passenger A had longer residence time that those exhaled by Passenger B and C. The longest particle residence time appeared in Passenger B’s own breathing zone could be as long as 10.2 s. Comparatively, the particles released by Passenger C are more easily carried on by the airflow. The computations also revealed the residence time of the particles in Passenger
B’s breathing zone when the particles were released by Passenger A, which was around 6.2 s. Such a long particle residence time indicated poor ventilation in Passenger B’s breathing zone; therefore, in order to minimize the risks thus caused, the personalized ventilation was strongly recommended for Passenger B.

5.4 Conclusions

This study investigated the effects of cough-jet on airflow field and droplets transport characteristics in a cabin section. Simulations are conducted by considering coughing as a transient generated process and one cough consists of different droplets size and number. Through comparing numerical results with experimental data, the prediction accuracy of the proposed numerical approach was validated. The conclusions yielded from this study are summarized as follows:

- Cough-jet has significant effects on air flow in front of cough-passenger in short period of time. In the time region of 0-0.5 second and the distance from mouth less than 0.45 meter, the cough-jet dominant the airflow. After this period, the effects of cough-jet gradually decrease, ended by the airflow come back to the steady state.
- Simulation results of droplet distribution by considering cough-jet model have a significant difference from without considering the cough-jet model. The most obvious difference was particles were carried away by ventilation flow immediately after released from mouth in without cough-jet model situation, while in applying the cough-jet model condition, particles were dominated by ventilation flow until the effects of cough-jet were decreased after a certain time and distance from mouth.
- Similar to a calm indoor environment (Zhu et al., 2006), the distribution process of droplet was affected by the size in the cabin. For particles with diameter larger than 5E-5m, gravity caused by self-weight dominated the particles motion, while drug force caused by airflow dominated the motion of particles with size smaller than 5E-5m.
Chapter 6

The Effects of Induced Pressure Fluctuations on Air Quality in HSR Cabins

The main findings of this chapter have been included in:


HSR trains usually experience alternating pressure waves when travelling through tunnels or crossing each other. The pressure waves can have a serious impact on vehicle stability and cause severe aural discomfort for passengers onboard. However, their effects on air quality and thermal comfort remain unknown. This study aims to answer this question through transient CFD simulations of a full-size CRH-2 HSR cabin. The response of cabin interior airflow, contaminant transport and thermal comforts to the transient pressure fluctuation during the period of two HSTs passing by each other were studied firstly. Then, the study of another operation condition, the HST
passing through a tunnel, was carried out. For the situation that two HSTs passing each other, the results indicate that the pressure fluctuation induced by two HSTs crossing each other is not a major consideration when investigating the thermal comfort and air quality in HSR cabins. For situation that HST passing through a tunnel, simulation results indicate that the induce pressure waves can influence the cabin ventilation performance, air quality and thermal comfort in the areas near the door. Also, the induced pressure wave had a strong influence on the passengers’ hearing pressure which lasts for the same period as passing through the tunnel.

6.1 High-speed Trains Passing Each Other

6.1.1 Introduction

The growing demands on high-efficiency transportation have motivated the development of high-speed rail (HSR) connecting major cities in many countries worldwide. By the end of 2016, the world had more than 40,000 kilometres of HSR (≥200 km/h) in operation and additional 20,000 kilometres under construction, carrying over 2 billion passengers annually. Like airliner cabins, HSR cabins are enclosed airtight spaces with high occupant density. The thermal environment and air quality in the cabins are of great importance to the ride comfort and public health.

Environmental control of HSR cabins is very challenging, not just because of the dynamic events induced by the frequent accelerating/braking actions of the vehicles, but due to the complexity resulted from the changing environmental and climatic conditions. Particularly, many advanced HSR trains (e.g., Chinese CRH, Japanese Shinkansen and French TVG) are running at speeds over 300 km/h, which can cause tremendous pressure waves (Ricco et al., 2007) when a train is travelling through a tunnel or two trains are crossing each other. Shown in Figure 6.1 are typical pressure waves induced by a HSR train travelling through a tunnel. According to the numerical study by Chu et al (Chu et al., 2014), the pressure waves could be as large as 10.0 kPa when two trains are crossing each other in a tunnel. Although HSR cabins are relatively isolated spaces from the exterior, the pressure waves still can intrude into the cabin through the air-conditioning system and carriage gaps, causing considerable pressure
fluctuations inside the cabin. The experimental tests by Liu (Liu, 2012) onboard a CHR-5 train revealed that the amplitude of pressure fluctuations in the cabin could be as large as 500 Pa within 2.5 seconds.

These pressure fluctuations have been reported causing discomfort, particularly aural discomfort (Palmero and Vardy, 2014), for the passengers on board. As a strategy to mitigate the interior pressure fluctuations, many HSR trains would automatically close the fresh air intakes and exhausts of the air-conditioning system when exterior pressure waves are detected (Liu, 2012). However, when the train is running on a mountainous terrain with a high density of tunnels, frequent shut-off of the intakes/exhausts can result in remarkably rising contaminant concentration and degraded air quality in the cabin, as experimentally proven by the field tests of Zhang and Li (2012) onboard a CRH-2 vehicle running from Guangzhou to Changsha (China). Although advanced control strategies that can provide adaptive pressure equalization (Liu, 2012) have been developed, it seems that the conflict between pressure comfort and air quality in HSR cabins remains an unsolved problem.

At the meantime, HSR cabins are densely occupied spaces with passengers approximately seated to each other. Huruya (Furuya, 2007) warned that train cabins could be a high-risk venue for the airborne transmission of infectious diseases such as
Tuberculosis and influenza. However, the transport and dispersion characteristics of airborne contaminants in fluctuating pressure fields have been rarely studied, which significantly impedes the assessment of infection risks in HSR cabins and development of precautions. As the extensive studies on airliner cabins (Liu and Chen, 2013, Zhang et al., 2009b) have proven that the dispersion of airborne contaminants in enclosed cabin spaces is predominantly controlled by the airflow field, a thorough understanding of the effects of pressure fluctuations on the thermal flow field and contaminant dispersion in HSR cabins is crucial to create a comfort and safe riding environment. Unfortunately, the important issue has long been overlooked. Therefore, this study, using numerical approaches, aims to achieve a quantitative analysis of the effects of the pressure fluctuations on the thermal environment and contaminant transport characteristics in HSR cabins.

6.1.2 CFD Models of Airflow Field in Densely Occupied Cabin Spaces

Owing to the relatively low computational cost and proven accuracy, the Reynolds-Averaged Navier-Stokes (RANS) equations have been the most popular theoretical model to analyse the thermal flow fields and air quality in enclosed indoor spaces. The RANS model can be expressed in the form of advection-diffusion equation of the generic scalar $\Phi$.

$$\frac{\partial}{\partial t}(\rho \Phi) + \nabla \cdot (\rho U \Phi - \Gamma_{\Phi} \nabla \Phi) = S_{\Phi}$$  \hspace{1cm} 6.1

where, $\rho$ and $U$ is the air density and velocity, respectively. $\Gamma_{\Phi}$ is the effective diffusivity and $S_{\Phi}$ is the source term of $\Phi$, respectively. By replacing $\Phi$ with a transportable scalar $C$ representing the concentration of contaminant in the air, Eq. 6.1 has also been widely used to model the transport of dilute gaseous and particulate contaminants in cabin spaces (Yan et al., 2016, Rai and Chen, 2012).

The mean age of air (MMA) was modelled using the transportable scalar $\tau$ (Fan et al., 2017). The transport equation takes a similar form to Eq. 6.1.
\[
\frac{\partial}{\partial t}(\rho \tau) + \nabla \cdot (\rho \mathbf{U} \tau - \Gamma \nabla \tau) = \rho
\]

As the age of air does not diffuse, only the turbulent diffusion was included in the diffusivity \( \Gamma \).

The above theoretical models, together with the RNG \( k-\varepsilon \) model for air turbulence (Chen, 1995a), have been widely validated for modelling air distribution in indoor spaces (Liu et al., 2012a). However, the airflow fields in HSR cabins have rarely been experimentally investigated, making it infeasible to directly validate the theoretical models based on realistic HSR cabin conditions. As HSR cabins are highly analogous to airliner cabins in terms of air distribution and seats layout, it is expected that the airflow pattern and contaminant dispersion characteristics in the cabins are comparable to each other. Therefore in this study, the models were compared against the experimental data of Li et al (Li et al., 2015a), which were acquired in a Boeing-737 aircraft cabin mock-up.

During the experiments (Li et al., 2015a), 42 electrically heated thermal manikins were seated in 7 rows of seats, with each row containing 6 manikins \(((3 + 3) \times 7 \text{ seats layout})\), as shown in Figure 6.2. Conditioned air was supplied through linear slots below the luggage compartment and exhausted through outlets close to the floor. The 2-dimensional (2D) airflow field in a vertical plane in front of the 4\(^{th}\)-row manikins had been measured using Particle Image Velocimetry (PIV) technique, which were used in this study for model validation. CFD simulation of the airflow field in the cabin mock-up was performed using the aforementioned theoretical models. Details of the boundary conditions and numerical procedures have been detailed one of publications (Yan et al., 2017) and will not be repeated here.
Figure 6.2 The cabin mock-up model for model validation (Yan et al., 2017)

Figure 6.3 shows a graphical comparison of the predicted air velocity vectors against the PIV data in the test plane, where the black vectors represent the PIV data and red vectors represent the numerical results. It was firstly found that two approximately symmetric vortexes were formed in the cabin, driven by the ventilation air jets. The numerical computation successfully captured the airflow pattern and vortexes. Although small deviations were detected in some local areas, they could be attributed to the different manikin geometry of this study to that used in the experiments (Li et al., 2015a). The quantitative comparisons of the predicted air velocity profiles along 6 vertical lines against the experimental data are shown in Figure 6.4. Again, it was found that the predicted air velocity agreed well with the experimental data along with all the 6 vertical lines.

Figure 6.3 Comparison of the predicted airflow vectors against the PIV image (Yan et al., 2017)
6.1.3 Effects of Pressure Fluctuations in HSR Cabins

6.1.3.1 The HSR Cabin Model

The validated models were then used to analyse the effects of pressure fluctuation in HSR cabins. A full-size HSR cabin model was built based on the actual layout of the Chinese CHR-2 2nd-Class passenger cabin, as shown in Figure 6.5. The cabin had dimensions of 19.2 m-length × 3.1 m-width × 2.25 m-height and accommodated 80 passengers ((3 + 2) × 16 arrangement). Conditioned air was supplied into the cabin through linear slot diffusers located immediately below the luggage racks and exhausted through the vents underneath the seats. Simple manikin models which were composed of rectangular blocks representing human body segments (head, torso and legs) were used in this study in order to save the computational cost. The surface area of each manikin was 1.50 m², which was close to the mean surface area of Asian females (1.522 m² according to the anthropometric data of Yu et al’s (Yu et al., 2010) ). However, the effective surface area for heat transfer of each manikin was 1.08 m², exclusive of the contacting area of the manikin with the seat. It was assumed a patient who was continuously releasing pathogen-carrying contaminants was seated close to the aisle, in the 8th row (Passenger 8C).
Two pathways through which the exterior pressure waves intrude into the cabin were considered in this study: the air-conditioning ducts and door gaps (Zhu et al., 2014). Depending on the air tightness of the cabin envelope and control strategy of the air-conditioning system, the amplitude of interior pressure fluctuations in actual HSR cabins can be distinctly different. For a typical CRH-2 vehicle running into a tunnel at a speed of 250 km/h, the pressure fluctuations measured at the exhaust vents and door gaps are shown in Figure 6.6 (Liu, 2012). It was found the interior pressure is slightly (66 Pa) higher than the atmospheric pressure in order drive the ventilation. As there is less resistance in the door gap pathway, through it the exterior pressure waves can generate more significant influence on the interior pressure (-490 Pa maximum) than through the air-conditioning ducts (-280 Pa maximum). The total time of pressure fluctuations was approximately 5.0 s, after which the pressure at the linear slots and
door gaps remained constant. In addition, due to the longitudinal transmission of the exterior pressure waves along the vehicle surface, the pressure curves at the front and rear door gaps had the same pattern but had a 0.25 s time difference.

Figure 6.6 Instantaneous pressure fluctuations measured onboard a CRH-2 cabin (Liu, 2012)

The computations were conducted under a summer scenario (ambient air temperature $T_a = 35 \, ^\circ C$). The ventilation parameters of the cabin were carefully calculated based on the Chinese Passenger Train Air-Conditioning Standard (TB 1951-87). A heat balance calculation was performed, in which the metabolic heat of passengers (89 W/person at a relaxing state (Yan et al., 2016)), ambient heat invasion and solar radiation were considered. The heat transfer from the ambient air into the cabin is calculated by

$$Q_w = hA(T_o - T_i)$$

where, $h$ is the integrated heat transfer coefficient of the cabin walls. $A$ is the effective heat transfer area. $T_o$ and $T_i$ are the outer and inner surface temperature of the cabin walls, respectively. When the solar radiation is taken into account, $T_o$ is estimated by
\[ T_u = T_e + \frac{\varepsilon J}{a_w} \]

where, \( \varepsilon \) is the absorption coefficient of wall material. \( J \) is the intensity of solar radiation and \( a_w \) is the thermal diffusivity of the wall material.

According to the cabin ventilation strategy, 70% of the exhausted air was returned back to the cabin and fresh air (25 m\(^3\)/person) only took 30%. When the target air temperature in the cabin was set at 27 °C, it was calculated that the mass flow rate and temperature of the supply air at the linear slot diffusers were 2.14 kg/s and 24 °C, respectively.

The computational domain was discretised using unstructured tetrahedral mesh with locally refined inflation mesh layers at the manikin surfaces. Mesh independence was achieved at 10.2 million mesh elements as a further increase to 11.3 million mesh elements only caused a negligible change (less than 0.5%) in the predicted air velocity profile along a randomly selected line. The equations were discretised based on the conservative finite volume method and solved using the commercial CFD code ANSYS-CFX 17.0. A steady-state computation that assumed the interior airflow field is free from the exterior pressure waves was firstly conducted to obtain the initial velocity, temperature and concentration fields. Then transient computations were performed to analyse the effects of pressure fluctuations on the interior fields. For the transient computations, variable time steps ranging from 0.1 s to 1.0 s were adaptively selected depending on the speed of pressure variation. It took 56 hours to simulate a 30-second transient process on a workstation desktop with 40 CPU cores.

### 6.1.3.2 Steady-state Indoor Environment in the HSR Cabin

6 planes were selected to analyse the cabin environment, as shown in Figure 6.5 Up. Plane X1, X2 and X3 are cross-sectional planes in front of the Row-1, Row-8 and Row-16 passengers, respectively. Plane Y1 is a horizontal plane at the nose level (1.1 m above the floor) of the passengers. Plane Z1 and Z2 are vertical planes cutting through the Column-C and Column-E passengers, respectively.
Figure 6.7 shows the air temperature contours and velocity vectors in the selected planes. It was firstly found that the air temperature and velocity fields present approximate periodicity along the cabin length (Figure 6.7(a) and (b)). The two streams of ventilation air injected from the linear slots collide with each other near the aisle, pushing the airflow downwards and driving two significant vortex flows in the cabin (Figure 6.7(c) - (e)). Due to the vortex flows, the thermal plumes of the passengers sitting near the aisle was suppressed (Figure 6.7(a)) while the thermal plumes of those sitting next to windows could be clearly seen (Figure 6.7(b)). However, due to the asymmetrical seat-arrangement, the vortex region in the 3-seat side is bigger than that in the 2-seat side, which is slightly different from that in the airliner cabin where the vortex regions are symmetrically distributed on both sides of the aisle (Figure 6.3).

Figure 6.7 also reveals that the CRH-2 cabin creates a comfortable thermal environment. Firstly, The average air temperature in the cabin was 300.0 K with the mean differential temperature between the head and ankle levels being only 0.4 K, which was significantly smaller than the 3 °C upper limit regulated by the ASHRAE 55-2004 Standard. Beyond that, the draught rating (DR, the number percentage of dissatisfied passengers due to draught) of the cabin was also calculated according to the
ISO Standards 7730 (ISO, 2005), based on the predicted air velocity and temperature fields.

\[ DR = (34 - T)(u - 0.05)^{0.62}(3.14 + 37u\xi) \]

where, \( T \) and \( u \) is the local air temperature and velocity, respectively. \( \xi \) is defined by \( \xi = 100(2k)^{0.5}/u \). It was found that the DR was 1.5% in this case, which was 10-folds smaller than the 15% permissible value recommended by the ISO Standards 7730.

The predicted mean vote (MPV) was also obtained for each passenger in the cabin based on the air velocity and temperature fields. The MPV here was calculated according to the ASHRAE 55 Standard (ASHRAE, 2004), which was based on the Fanger model. Figure 6.8 shows the MPV map in the cabin. It is clear that the CRH-2 vehicle provides a comfortable thermal environment as the MPV values sit in the range of -0.5 to 0.5 (ASHRAE, 2004). Comparatively, the Column-C passengers feel a bit colder than the others as they are subject to the ventilation jets (Figure 6.7).

Figure 6.8 Map of the predicted mean vote (PMV) for each passenger

The predicted mean age of air (MMA) in the breath plane (Plane Y1) is shown in Figure 6.9. It shows that the aisle region has the most portion of fresh air, which is consistent with Figure 6.7. In addition, the asymmetrical seat-arrangement has a detectable effect on the MMA distribution in the cabin. The MMA in the 3-seat side
was significantly larger on the other side of the aisle. Particularly, due to the lock-up effect of the vortexes, the MMA in front of the passengers was apparently prolonged. The vortexes have been experimentally found to be able to lock up CO$_2$ exhaled by the passengers in an airliner cabin. Therefore, personalized ventilation seems essential in order to pursue improved air quality in the passengers’ breathing zone. The Air change effectiveness (ACE) was calculated based on Figure 6.9, using Eq. 6.6.

$$ACE = \frac{\tau_n}{\tau_{ave}}$$  \hspace{1cm} \text{(6.6)}$$

where, the nominal time constant $\tau_n$ is the ratio of the air volume of the cabin to the rate of air supply. It was calculated that the ACE of this case was 99.4%, which was significantly superior to the 0.95 lower limit regulated by the ASHRAE 127-1997 Standard.

![Mean age of air (MAA) at Plane Y1](image)

**Figure 6.9 Mean age of air (MAA) at Plane Y1**

The predicted distribution of contaminant concentration in Plane Y1 is shown in Figure 6.10, where the concentration was normalized in terms of the releasing concentration at the index patient’s surface and was illustrated using a logarithmic colour scale. It shows that the contaminant was mainly concentrated in a small area around the index patient. Its concentration decreases quickly with increasing distance from the index patient. However, low concentration of contaminant was not found near the aisle, which was significantly different from the MAA pattern in the same plane.
6.1.3.3 Effects of Pressure Fluctuations

The transient computations revealed that the pressure in the cabin responded quickly to the exterior pressure waves. Figure 6.11 shows the transient pressure measured near the ears of Passenger 1C and 8C, respectively. A comparison of Figure 6.10 against Figure 6.6 reveals that the interior pressure is almost synchronous with the exterior pressure, indicating the fact that the pressure transmission is very prompt. It also indicates that the pressure fluctuations can cause immediate aural discomfort for the passengers, as discovered by many studies (Palmero and Vardy, 2014).

However, the instantaneous pressure field in the cabin was found to keep homogeneous during the entire pressure fluctuation process as only very small differential pressure was detected at a given time step. Typically, shown in Figure 6.12 is the pressure...
contour map in Plane Z1 at $t = 2.0s$. Although the pattern of pressure wave march can be clearly seen, the maximum pressure difference in the cabin was only 12 Pa. Due to its small magnitude, this instantaneous differential pressure was not expected to have significant effects on the airflow field. In order to quantify the effects of pressure fluctuation on the thermal flow field in the cabin, the air velocity and temperature profiles along three vertical lines (Line 1, 2 and 3) which are intersected by Plane Z1 respectively with X1, X2 and X3 (see Figure 6.5 Up), were shown in Figure 6.13 Up and Down, respectively. The figures show that the air velocity and temperature profiles along Line 1 and 2 were only negligibly changed by the pressure fluctuations. Only those along Line 3, which was very close (0.15 m) to the door gap, were slightly affected. The increased negative pressure at the door gap caused a gust of air accelerating towards the outside. However, the air temperature was not changed due to the outwards gust. Similarly, the effects of pressure fluctuation on the contaminant concentration were also negligible due to its weak effects on the airflow field, as shown in Figure 6.14.

![Air Pressure Contour](image)

*Figure 6.12 The air pressure contour in Plane Z1 at $t = 2.0 \, s$*
Figure 6.13 Effects pressure fluctuation on the thermal flow field in the cabin: Air velocity (Up); Air temperature (Down)

Figure 6.14 The effects of pressure fluctuation on the contaminant concentration field were negligibly small
The effects of the pressure fluctuation on the PMV were also analysed. The local PMV values of Passenger 1C and 8C are shown in Figure 6.15. It was predicted that both passengers experienced a transitory PMV decrease. The PMV regained their original levels in around 10 seconds. Comparatively, Passenger 1C experience more distinct PMV drop as it was seated close to the door gap. However, considering the short time duration, it was expected that the passengers even cannot physically feel the thermal condition changes.

![Figure 6.15 The influence of pressure fluctuation on the PMV of passengers](image)

### 6.1.4 Conclusions

Transient numerical simulations were conducted in this study to analyse the effects of pressure fluctuations induced when two HSTs crossing each other on the thermal environment and air quality in a CHR-2 HSR cabin. Two major pathways through which the exterior pressure waves intrude into to cabin, namely the door gaps and air-conditioning ducts, were considered. The computations revealed that the interior pressure field responded instantly to the exterior pressure change and can cause significant aural discomfort. However, the pressure fluctuation, despite its considerable magnitude, cannot build a pressure gradient that is large enough to change the airflow pattern in the cabin. As a result, neither the thermal flow field nor the contaminant dispersion pattern was significantly influenced by the instantaneous pressure fluctuations. It indicates that the pressure fluctuation induced by two HSTs crossing
each other is not a major consideration when investigating the thermal comfort and air quality in HSR cabins.

### 6.2 High-speed Train Passing Through Tunnels

#### 6.2.1 Introduction

During the past decades, High-speed trains (HSTs) are rapidly developed in many countries with the operation speed increased from 200 km/h to 500 km/h (Luo, 2016). The speed-up train system has attracted many researchers' interests on aerodynamic problems due to the impact of strong disturbance on the surrounding air (Zhao et al., 2013). Particularly, when two HSTs are passing each other, the disturbance will be aggravated, which would induce very large transient pressure fluctuations. Li et al. (2016c)'s research found that the pressure wave on the cabin outside surface could influence running security and passenger comfort. Tian et al. studied the HSTs passing behaviours at different speed levels and found that pressure wave was proportional to the train speed. Raghunathan et al. (2002) also demonstrated that the induced pressure wave was one of the most important limitations to HST speed-up.

Most recent studies of induced pressure wave focus on the pressure wave acting on the outside surface of HST such as the aerodynamic and aeroacoustic performances. However, according to Liu and Li (2012), the induced pressure wave could also influence the cabin inside air environment through cabin ventilation system. The exhaust vents of air conditioners were exposed to the outside surface of train cabin. As the cabin inside air environment was mainly dominated by the air conditioning system (Xu et al., 2013), the induced pressure wave could indirectly affect the cabin inside air environment. In addition, Zhang and Li (2012) found that the outside pressure could also affect the inside cabin environment through the cabin connection gaps. Their research found that the back-door-gap-oriented design of exhaust outlets in CRH3 model cabin had strong ability in removing cough droplets.

Therefore, from previous researches, the induced pressure fluctuation could impact on cabin interior airflow environment. The main reason that not many studies issued this
topic were due to the extremely high computational cost. According to Gupta et al. (2010)’s simulation case, a 4-min 7-row cabin ventilation transient simulation case took 4 weeks to run using an 8-parallel-processor computer cluster. Even the induced pressure fluctuation happens in less than 20 seconds (Liu and Li, 2012), the computational cost is very high because of the complicated HST cabin geometry and fully occupied passengers. Therefore, the necessity of applying the induced pressure wave in cabin interior ventilation simulation needs to be assessed.

In this study, the inside cabin ventilation performance and airflow response in the whole HST carriage were numerically investigated using Computational fluid dynamics (CFD). Emphasis was the effects of induced pressure fluctuation on the airflow field, contaminant transport and passenger ride comfort. CFD models of a typical HST cabin were firstly developed and validated using experimental data yielded from previous study (Yang et al., 2017). During the period that HST passing through a 1 km tunnel, the exterior pressure wave could intrude into the cabin through the door-daps and the air-conditioning vents (Liu and Li, 2012). The ventilation response in a short period of time was assessed in different time-steps. The distribution process of gaseous contaminant released from a passenger located in the middle of the cabin was studied. Also, the passenger ride comfort including Predicted Mean Vote (PMV) were investigated.

6.2.2 Methods

Considering that the high occupation in an economy-class cabin may cause a high risk of contaminant transmission, this study focuses on the inside-cabin air-quality of a fully occupied economy-class cabin. Because the induced pressure wave directly influences the pressure at the cabin connection gaps, a whole HST cabin model was applied. In order to minimise the geometry complexity, the luggage-store area and the restroom area in a normal HST cabin were not considered. As illustrated in Figure 6.16, a 16-row CRH2 model cabin with 3-2 seat arrangement was set up as the ventilation environment. In this cabin, all the seats were occupied with 80 simplified manikin models.
The ventilation schema was built based on the frame of the prevalent railway HVAC standards UIC 553-1 (UIC, 2005). In this study, it assumed that the HST was operating in summer when the outside temperature was around 35°C. According to the UIC 553-1 standard, the air supply rate through the inlets was 2.14kg/s. Each occupied manikin surface was set up with a convective heat load of 40W, which could imitate the passengers’ body heat (Salmanzadeh et al., 2012). Also, due to solar radiation effects, an equivalent heat conduction rate of 688.5 W/m² was set through cabin roof, walls and windows. The rest of solid walls, including cabin floor and seats, were assumed to be at the adiabatic condition. Considering the energy conservation law, the thermal balance in the cabin can be expressed as follows:

\[ Q_{\text{air}} = Q_{\text{wall}} + Q_{\text{human}} \]  

6.7

Which means the thermal energy generated and absorbed inside a cabin is equal to the energy taken away by the airflow from inlet vents to outlet vents (\( Q_{\text{air}} \)). The main sources of thermal load inside the cabin are the heat flux of human body (\( Q_{\text{human}} \)) and thermal conduction of cabin wall (\( Q_{\text{wall}} \)). Based on the above calculation, the inlet air temperature was set as 24°C.

According to the field measurement by Liu et al. (2017b), the induced pressure fluctuations during the HSTs passing through a 1km tunnel events in the speed of 300 km/s were plotted in Figure 6.17. From this figure, both induced pressure fluctuations of exterior cabin and interior cabins have similar change trends. Two pressure peaks
could be observed for both waves. However, the magnitude of interior pressure wave is much smaller than the exterior pressure wave. The overall equivalent induced pressure wave lasted less than 21s and the pressure change ranges are 3000Pa for exterior pressure wave and 900Pa for interior pressure wave. In the simulation model, the exterior pressure wave intrusion was considered through the door-gaps and air-conditioning vents into the HST cabin. According to (Liu and Li, 2012), the pressure wave at door-gaps is almost the same as the pressure induced at the cabin outside surface. Considering the 300km/s relative travelling speed of HSTs when passing through the tunnel, the pressure at the back-door-gap was induced 0.2s later than it at the front-door-gap. Also, because of the air-conditioning compressor, the equivalent pressure wave at ventilation outlet was much smaller than it at the cabin outside surface.

![Figure 6.17. Induced pressure fluctuations measured when a HST passing through a 1km tunnel (Liu, Chen 2017)](image)

As RANs model was widely used for solving the airflow field in the indoor environment with a relative low computational cost, RANS equations were applied for solving the airflow and air quality in this study, as shown below:

\[
\frac{\partial \phi}{\partial t} + \frac{\partial}{\partial x_i} \left( u_i \phi - \Gamma \frac{\partial \phi}{\partial x_i} \right) = S_\phi
\]

6.8

Where \( \phi \) is the general variable; \( u \) is the fluid velocity; \( \Gamma \) is diffusion coefficient; \( S_\phi \) is the source term.
Accompany with RANs model, RNG k-epsilon model was used for modelling the turbulent feature of the airflow inside the cabin. This method was validated by Yan et al. (2016). Boussinesq approximation was involved to address thermal buoyancy flow induced by human body heat.

For gaseous contaminants, acetaldehyde, a typical human-released VOC in gaseous contaminant sector (Mochalski et al., 2014), was assumed to be released from one passenger’s surface. Based on the Ansys CFX-Solver Theory Guide (ANSYS, 2016), for the extremely low concentration of gaseous contaminant, the prediction of distribution could be tracked accurately by using the drift-flux model:

$$\frac{\partial \phi}{\partial t} + \frac{\partial}{\partial x_i} (U \phi) = \frac{\partial}{\partial x_i} \left( D \frac{\partial \phi}{\partial x_i} \right) + S_\phi$$

Where $U$ is the fluid velocity in air-flow domain; $\phi$ is the concentration of contaminant; $S_\phi$ is a volumetric source term; $D_\phi$ is the kinematic diffusivity.

The cabin geometry model was discretised using unstructured mesh by ANSYS ICEM. For capturing accurate features of the induced pressure wave, high resolution meshes were built around ventilation outlet and door-gaps. Also, inflation layers with an exponential growth ratio were applied around manikin surfaces. Prior to CFD simulations, grid independency test was conducted, which found that as long as the grid element number was bigger than 10.2 million, airflow velocity at the reference location became stable.

The above-mentioned theoretical models have been validated using experimental data from a cabin mock-up. The validation results revealed that the proposed numerical model was valid in simulating the HST cabin ventilation. Details of validation process could be referred to authors’ previous study (Yang et al., 2017). For validation the pressure intrusion process, the pressure gradient of interior cabin from experiment by Liu et al. (2017b) and simulation results have been compared in Figure 6.18. It is clear that both experiment and simulation results have similar pressure change trends for the
18 seconds. Thus, the pressure intrusion process for the simulation has been validated as accuracy.

![Figure 6.18 Pressure gradient from experimental and simulation results](image)

All the theoretical models were discretised based on conservative finite volume method and solved using software ANSYS CFX 17.0. Because the occurrence of induced pressure wave was an unsteady process, transient simulation was conducted after the steady-state simulation. According to Figure 6.17, the equivalent pressure wave was changed in very high ratios. Therefore, in the transient simulation, the real-time was dived into small computational time-steps so that high accuracy of airflow and containment transport can be achieved. Especially for the period of large pressure changing section, such as between 1-2 second, the transient computational time-step was set as 0.05-second. Overall, 18-second transient simulation case cost 5 days to run on a 40-parallel-processor computer cluster.

### 6.2.3 Results and Discussion

The object of this study was to investigate the influence of induced pressure wave during HST passing through a 1km tunnel on cabin interior air environment. Thus, this project would discussion in three aspects: the ventilation performance, containments distribution and ride comfort.
6.2.3.1 Ventilation Performance

The mean temperature and velocity through the whole train cabin at different time-steps during the HST passing through the tunnel were compared to describe the ventilation performance, as shown in Figure 6.19. In the steady state, when \( t=0 \)s, the mean temperature is near 299.00K, which is under the designed ventilation requirements. After train entering the tunnel, the mean temperature increased into 299.07K at 3 second. Then, after 4 seconds slightly reducing, the cabin mean-temperate changed from 299.03K to 299.11K. In the whole passing period, the inside temperate change is very small. Figure 6.19 illustrate the mean cabin temperature has an overall increment trend. It is caused by the intruded exterior airflow, which is warmer than the interior temperature. Overall, the induced pressure wave has very slightly influence on the whole cabin interior temperature.

Unlike the mean temperature has an overall increasing trend, the mean velocity inside the cabin increased at first 3 second, and then reduced in for 6 seconds. The mean velocity increment at 3 second reached 0.33 m/s, which is 26% higher than the initial mean velocity. At 9 seconds, the mean velocity reached the lowest speed of 0.22 m/s. Small velocity fluctuation could be found after 9 seconds until passing through the tunnel. Therefore, except the quick velocity change in the first 6 seconds, the overall velocity change during HST passing through 1km tunnel is not obvious.
Figure 6.19 Mean temperature (Up) and velocity (Down) profiles at a selected plane in different time-steps

To better understand the influence cabin area by the intruded pressure wave, Figure 6.20 presents the temperature and velocity on one line above manikins seated in the third row at different time steps including 2, 6, 8, 10 and 15 second. Very well matched periodic temperate and velocity changing patterns could be found in the x-position between 4 - 16m area. However, in the location between 0 - 4m (near the front door), and the location between 16 - 20m (near the back door), the temperature and velocity changes were not consistent at different time steps. Thus, the influence distance of pressure fluctuation for temperature and velocity could be identified as around 4m near the front and back doors.
6.2.3.2 Containments Distribution

The distribution characteristic of gaseous contaminant released by both passenger 8C and passenger 2C was studied to evaluate the effects of induced pressure wave on transporting occupant-generated pollutants. VOCs concentration rendering (Figure 6.21) indicates the distribution performance of VOCs in steady-state. The normalized concentration of contaminant was also marked in this figure. Overviewing, VOCs only concentrate around neighbourhoods of the source passenger. Table 6.1 illustrated the VOC relative concentration ratio in high concentration positions, the P8B (seat next to source manikin P8C) and the P2B (seat next to source manikin P2C) under different time-steps during HST passing through the 1km tunnel. The VOC concentration for P8B was changing in a small fluctuation, which is less than 10%. Compared with the large fluctuation of pressure, this small variance could also be neglected. However, the

Figure 6.20 Temperature (Up) and velocity (Down) above manikins seated in the third row at different time steps.
fluctuation of VOC concentration for P2B was 50% higher than the fluctuation for P8B. Therefore, the induced pressure wave has much a stronger effect on the concentration of gaseous contaminant for the area near the door.

Figure 6.21 VOCs concentration rendering around released passenger in steady-state

Table 6.1 VOC relative concentration ratio in high concentration positions: P8B (seat next to source manikin) and P7C (seat in front of source manikin)

<table>
<thead>
<tr>
<th>Traveling time (s):</th>
<th>0 (Before Entering)</th>
<th>1</th>
<th>2</th>
<th>3</th>
<th>4</th>
<th>5</th>
<th>6</th>
<th>7</th>
<th>8</th>
<th>9</th>
</tr>
</thead>
<tbody>
<tr>
<td>P8B (%):</td>
<td>3.66</td>
<td>3.61</td>
<td>3.69</td>
<td>3.57</td>
<td>3.72</td>
<td>3.83</td>
<td>3.89</td>
<td>3.86</td>
<td>3.77</td>
<td>3.69</td>
</tr>
<tr>
<td>P2B (%):</td>
<td>3.54</td>
<td>3.50</td>
<td>3.52</td>
<td>3.44</td>
<td>3.82</td>
<td>3.97</td>
<td>4.12</td>
<td>4.06</td>
<td>3.93</td>
<td>3.81</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Traveling time (s):</th>
<th>10</th>
<th>11</th>
<th>12</th>
<th>13</th>
<th>14</th>
<th>15</th>
<th>16</th>
<th>17</th>
<th>18</th>
<th>19</th>
</tr>
</thead>
<tbody>
<tr>
<td>P8B (%):</td>
<td>3.59</td>
<td>3.61</td>
<td>3.66</td>
<td>3.60</td>
<td>3.44</td>
<td>3.36</td>
<td>3.44</td>
<td>3.56</td>
<td>3.63</td>
<td>3.66</td>
</tr>
<tr>
<td>P2B (%):</td>
<td>3.54</td>
<td>3.42</td>
<td>3.45</td>
<td>3.40</td>
<td>3.26</td>
<td>3.11</td>
<td>3.27</td>
<td>3.41</td>
<td>3.50</td>
<td>3.56</td>
</tr>
</tbody>
</table>
6.2.3.3 Ride Comfort

In order to quantify the ventilation performance, PMV, the most-recognized index for thermal comfort (Gilani et al., 2015), was calculated based on the Fanger’s heat balance equation (ASHRAE, 2004). The changing range of PMV value on each passenger during HST passing through 1 km tunnel were plotted in Figure 6.22. It can be seen from this figure that the changing range of each passenger’s PMV value is quite different. For the passengers seated near front door have the largest change range, while the passengers seated in the middle area of cabin have relatively small change range. The range of PMV fluctuation reached more than 10% for those who seated near front door. Overall, the thermal comfort could only influence the passenger seat near door by the pressure wave.

Figure 6.22 The changing range of PMV value for each passenger during the 18 seconds

Figure 6.23 shows the average pressure measured near the passengers’ ears during HST passing through the 1 km tunnel. Unlike the containment concentration or the thermal comfort, the hearing pressure had very strong response to the exterior pressure wave. The maximum negative hearing pressure value was collected when t= 3s, while the maximum negative value of the induced pressure wave was collected at t=16s. A
response delay between induced pressure waves and hearing pressure could be observed which around 0.8 second. Overall, the hearing pressure had the similar change pattern with the induced pressure wave. The hearing pressure changing ranges were around 500 Pa. After 17 seconds, the hearing pressure changed back to normal. According to (Raghunathan et al., 2002), the hearing pressure fluctuations could cause immediate aural uncomfortableness for passengers. Therefore, the induced pressure wave has strong influence on the hearing pressure, which may cause ride uncomfortableness for 17 seconds.

![Graph of hearing pressure over time](image)

*Figure 6.23 Average hearing pressure during the 18 seconds*

### 6.2.4 Conclusions

By building the transient numerical model of the ventilation system in a HST cabin, the influences of the induced pressure wave on cabin airflow, contaminant transport and passenger ride comfort were investigated. A transient simulation case was conducted with exterior induced pressure wave intrude into cabin when passing through a 1km tunnel. Simulation results indicate that induced pressure wave has minor effects on the average temperature and velocity in the whole cabin. But obvious airflow changes could be found in the area near front and back door. As the result, in the area near door, both thermal comfort and contaminant transport can be affected by the induced pressure wave. Also, the induced pressure wave had strong influence on the passengers’ hearing pressure which lasts for the same period as passing through tunnel. This study demonstrates that for the scenario that the HST passing through a 1km tunnel, the
induce pressure wave may only influence the cabin ventilation performance, air quality and thermal comfort in the area near door. Thus, it is not a major consideration for the rest area of train cabin. However, as the duration of pressure wave only lasting around 18-second, it is not clear in the situation that if the duration of pressure wave lasting much longer, such as HST passing through a longer tunnel or several continuous tunnels. Therefore, it is recommended to investigate the influence of long-duration pressure wave on ride comfort and contaminant distribution in the further work.
Chapter 7
The Effects of Solar Radiation on Thermal Comfort in HSR Cabins

The main findings of this chapter have been included in:


With the fast development of high-speed rail (HSR) around the world, high-speed trains (HSTs) are becoming a strong competitor against airliners in terms of long-distance travel. Compared with airliner cabins, HST cabins have much larger window sizes. When the big windows provide better lighting and view of the scenery, they also have significant effects on the thermal conditions in the cabins due to the solar radiation through them. This study presents a numerical study on the solar radiation on the thermal comfort in a typical HST cabin. The effect of solar radiation was discussed in terms of airflow pattern, temperature distribution, and thermal comfort indices. Parametric studies with 7 different daytime hours were carried out. The effects of using the roller curtain were also studied. It was found that overall cabin air temperature, especially near passengers, significantly increases with the presence of solar radiation. Passengers sitting next to windows had an obvious thermal comfort variation at
different hours of the day. To improve the ride comfort and reduce energy consumption in hot weather, using a curtain could effectively reduce the solar radiation effects in the cabin environment.

7.1 Introduction

In recent years, HSRs are rapidly developed in many counties due to convenience and high efficiency. A growing number of people choose HST other than airliners nowadays as the way of long-distance travelling (Albalate et al., 2015). During a long-distance travelling, a comfortable thermal environment is critical for passengers’ health and well-being (Zhang et al., 2009a). In addition, the fatigue of passengers after long journeys could diminish to a great extent (Alahmer et al., 2011). Therefore, maintaining thermal comfort in HSTs has become the main design task for HVAC designers and system developers.

According to a study of Simion et al (Simion et al., 2016), solar radiation is one of the crucial issues in ventilation system design. It can significantly affect ventilation system performance in enclosed space. In the past decade, a number of simulative and experimental studies have investigated the effect of solar radiation on HVAC and thermal comfort in the vehicular environment. By comparing the simulative and experimental results, Neacsu (Neacsu et al., 2017) found that the sun position has strong influence on cabin temperature and passenger’s comfort. Lee (Lee et al., 2014) pointed out that the solar radiation could heat up both exterior and interior surface of a car and consequently change the thermal comfort. Based on different simulative results of numerical radiation models, Moo (Moon et al., 2016) demonstrated that the spectral radiation effect must be considered in CFD simulation for accurate car cabin air temperature prediction. A 1-2°C temperature increment has been found after involving the radiation model in the simulation. All these studies revealed the fact that solar radiation plays important roles in ventilation performance and passengers’ thermal comfort in passenger compartments.

However, studies on the effect of solar radiation on thermal comfort in HST cabins are rare to see. Unlike metros which can be operated in tunnels or on shade ways, HSTs are
usually operated in the open air where they are directly exposed to the solar load (Raghunathan et al., 2002). Thus, HSTs are more likely to suffer solar radiation. Additionally, the process of radiative rays transmitting through windows into HST cabin is complex, which involved many impact factors. On one hand, the solar radiation condition is based on a dynamic environment, which related to solar emitting angle, radiation intense, local time, location coordinates, and weather condition (Ahmad and Tiwari, 2011, Cook et al., 2008). On the other hand, the HST cabin itself could affect the results of solar radiation, such as the train travelling direction, the material of cabin and window, the arrangement of passengers, and cabin geometry (Bouvard et al., 2018). Therefore, considering the specific characters of HST cabin and the complexity of solar radiation process, it is necessary to access the thermal comfort in HST cabins under solar radiation environment.

The main objective of this study was to investigate the solar radiation effect on thermal comfort in a typical HST cabin. Computational fluid dynamics (CFD) models and numerical procedures for HST cabin simulations were developed. Based on previous simulation and experimental results (Rodler et al., 2015, Gendelis and Jakovics, 2008), the solar radiation model was numerically examined. The typical thermal comfort index models, Fanger’s model and Percentage Dissatisfied (PD) Due to Draft model, were coupled with the CFD simulation to evaluate the thermal environment around passengers. Firstly, the simulation results considering solar radiation were compared with the case not considering radiation. The sensitivity parameters of solar radiation, including different time stages, solar load intensities and radiation angles, were also highlighted in the discussion. In addition, the impacts of using roller curtains which changes the window transmissivity rate of solar ray were discussed in the last section.

### 7.2 Numerical Simulation

#### 7.2.1 Model Description

In this study, a 2nd-class CRH2 passenger cabin, a typical HST cabin in China, was chosen as the geometrical model. The cabin geometry model considered key design parameters which are influential to the performance of the HVAC system (Zhu et al.,
Such parameters include cabin dimensions, type of diffusers, location of exhaust vents and seat arrangement. Figure 7.1 illustrates the computational model of a 16-row CRH2 cabin. The cabin dimensions are 19.2 m-length × 3.1 m-width × 2.25 m-height and it has a 3-2 seat arrangement. To minimize the geometric complexity, the bulk-luggage storage room and the restrooms were not included in this model. Ventilation air was supplied from 32 linear diffusers located on each side of the cabin wall, beneath the luggage rack and was exhausted from 32 outlets located under the seats.

![A typical HST cabin model](image)

*Figure 7.1 A typical HST cabin model*

For meshing the HST cabin model, the ANSYS ICEM CFD 17.2 software was used. Due to the complexity of detailed features of the manikin, triangular elements were used on the manikin surface and unstructured tetrahedral grids were adopted in the airflow domain. Prior to the primary CFD simulation, a meshing independency test was conducted by comparing the numerical solutions for the velocity using different mesh numbers. For different mesh densities, the velocity profiles near a randomly selected manikin are plotted in Figure 7.2. It can be seen that the simulation results were meshing-independent when the number of mesh elements exceeded 8.6 million.
Figure 7.2 Mesh independence test

In order to evaluate local thermal characteristics of each passenger, the cabin was fully occupied with 3D-scanned adult manikin models. The net surface area for heat transfer of each manikin was 1.02 m² exclusive of the manikin-seat contacting area. The passengers were assumed to be clothed with light-colored clothes in early autumn. A uniform and constant 60 W/m² heating power were applied at each manikin surface to imitate the human body heat (Melikov, 2015).

On each side wall, 8 double-glazed windows were evenly distributed along the cabin lengthwise direction. The dimension of each window is 1.8 m in width and 0.8 m in height. The total window surface area was 23.0 m², and the remaining cabin wall surface was 220.1 m². The interior volume of the HST cabin was 128.7 m³. The material properties specified for the cabin compartment surface and the manikins are listed in Table 7.1.

Table 7.1 Material Properties of the HST cabin and passenger

<table>
<thead>
<tr>
<th>Material</th>
<th>Density (kg/m³)</th>
<th>Specific heat (J/kg·K)</th>
<th>Thermal conductivity (W/m·K)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Surface</td>
<td>Material</td>
<td>Thickness</td>
<td>Density</td>
</tr>
<tr>
<td>-----------------</td>
<td>---------------------</td>
<td>-----------</td>
<td>-----------</td>
</tr>
<tr>
<td>Passenger</td>
<td>Cotton</td>
<td></td>
<td>1297.01</td>
</tr>
<tr>
<td>Cabin wall</td>
<td>ABS plastic</td>
<td></td>
<td>996.35</td>
</tr>
<tr>
<td>Cabin floor</td>
<td>Carpet</td>
<td></td>
<td>1601.85</td>
</tr>
<tr>
<td>Seat</td>
<td>Polyurethane foam</td>
<td></td>
<td>70.00</td>
</tr>
<tr>
<td>Window</td>
<td>Double-glaze (incl.</td>
<td></td>
<td>2529.58</td>
</tr>
<tr>
<td></td>
<td>roller curtain</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

The ventilation parameters were carefully calculated based on the prevalent railway HVAC standards UIC 553-1 (UIC, 2005). In the simulation, it was assumed that the HST was operated in early autumn in Shanghai and the outside temperature was 35°C. Heat conduction across cabin roof and walls was estimated using an integrated heat transfer coefficient of 2.22W/m²-K (Xu et al., 2011). The rest of solid walls including seats were assumed to be adiabatic. According to the cabin ventilation strategy (MRPRC, 1987), 70% of the exhausted air was returned to the cabin and the other 30% was fresh air (25 m³/person). For the whole train cabin, it is assumed that the total thermal energy put into the cabin is equal to the energy taken away by the ventilation airflow (Q_air). The main sources of thermal load inside the cabin are the heat flux of human body (Q_human) and thermal conduction of cabin wall (Q_wall). Therefore, considering the energy conservation law, the thermal balance in the cabin can be expressed as following:

\[ Q_{air} = Q_{wall} + Q_{human} \]  

7.1
Thus, according to the preformed heat balance calculation, the inlet air temperature was set to 24°C, when the air supply rate through the inlets was 2.14 kg/s.

7.2.2 Governing Equations

The cabin thermal flow field was solved using the incompressible Navier–Stokes equations in conjunction with the Boussinesq approximation for the buoyancy flows and the RNG k-ε model for the air turbulence. These models have been well validated by many studies (Yan et al., 2016) and proven to be robust for modelling indoor thermal flows. The model equations have been detailed in previous publication and will not be expatiated here.

To model the radiative heat transfer, the HST cabin was considered as an enclosure of grey-diffuse surfaces. The radiative exchanges between these surfaces were calculated using the Surface-to-Surface (S2S) model (ANSYS, 2016). In this model, the energy flux leaving a given surface is composed of direct emitted and reflected energy:

\[ q_{out,k} = \varepsilon_k \sigma T_k^4 + \rho_k q_{in,k} \] \hspace{1cm} \text{(7.2)}

where, \( q_{out,k} \) is the energy flux leaving the surface \( k \), \( \varepsilon_k \) is the emissivity of surface \( k \), \( \sigma \) is the Stefan-Boltzmann constant, \( T_k^4 \) is the temperature of surface \( k \), \( \rho_k \) is the density of surface \( k \), and \( q_{in,k} \) is the energy flux incident on the surface \( k \) from surroundings.

In the S2S model, the amount of incident energy upon the surface \( k \) from another surface \( j \) depends on the surface size, separation distances, and orientations. These parameters are accounted by a geometric function, the view-factor \( F_{kj} \), which is the fraction of energy leaving surface \( j \) that is incident on surface \( k \). Thus, the energy exchange between two surfaces \( j \) and \( k \) is expressed as

\[ q_{in,k} = \sum_{j=1}^{N} F_{kj} q_{out,j} \] \hspace{1cm} \text{(7.3)}
In this equation, the view-factor is calculated using the integration method (ANSYS, 2016). For randomly selected two surfaces \( j \) and \( k \) in 3.32, the view-factor \( F_{kj} \) can be calculated by

\[
F_{kj} = \frac{1}{A_k} \int_{A_j} \frac{\cos \theta_k \cos \theta_j}{\pi r^2} \delta_{kj} dA_j dA_k
\]

where, the parameter \( \delta_{kj} = 1 \) when \( dA_j \) is visible to \( dA_k \) and otherwise \( \delta_{kj} = 0 \). As the geometry in HST cabin is complex and the total mesh number achieved 8.6million, the integration method for radiation calculation process can be computationally very costly. To reduce the computational cost, the Surface-Clustering technique (ANSYS, 2016) was employed, which, instead of calculating each surface grid element, calculates the mean view factor and surface temperature of a cluster of surface grid elements. This method, although being efficient in reducing the computational cost, may result in significant inaccuracy when the clustering number is improperly chosen. Here, the surface cluster independency study was conducted to choosing the number of mesh elements in each cluster. The result of surface cluster sensitivity study was shown in Figure 7.3. It illustrates that once the number of faces per cluster reduced to a number less than 10, temperature at the reference location became stable. Therefore, the 10 mesh faces per surface cluster was set up in the simulation.

![Figure 7.3 Surface cluster sensitivity test](image-url)
To account for the radiative effects on the thermal flow field, the radiative heat transfer was jointly solved with the airflow field. To do this, the incident heat flux (Eq. 7.2) was incorporated in the CFD model as a boundary condition of the energy equation. The local energy balance of the grid element $k$ on the solid surface was expressed by

$$q_{ext} + q_{in,k} - (1 - \varepsilon_k)q_{in,k} - h_k(T_k - T_a) - \varepsilon_k\sigma T_k^4 = 0$$

where, $q_{ext}$ is the flux of externally applied heat, $h_k$ is the convective heat transfer coefficient and $T_a$ is the air temperature. The items on the left-hand side of Equation 7.5 represents the externally applied, incident, reflected, convective and directly emitted heat flux components, respectively.

For the numerical calculation, all the governing equations were solved using the commercial CFD software ANSYS Fluent 17.0. Each presented simulation case took approximately 6-hour computational time when using an 8-parallel-thread 32GB RAM computer.

### 7.2.3 Solar Load and Boundary Wall Details

As aforementioned, solar load acting on the HST cabin is one of the key inputs of thermal comfort analysis. When the solar ray entered the HST cabin domain, the solar-load model (ANSYS, 2016) was used to calculate the radiation effects of the solar ray. Both direct and diffuse solar irradiations were considered as the illumination parameters. In the solar-load model, the Solar-Calculator (ANSYS, 2016) was applied to obtain the incident angle of solar relative to the HST cabin and illumination parameters of the solar load with a specific location, date, time, weather condition and HST cabin’s moving direction.

As the HST in this work was assumed to be operated in early autumn in Shanghai, the longitude and latitude was set to $121.322^\circ$E and $31.19^\circ$N respectively. Simulations were conducted at different times of a day from 10:00 to 16:00 with one-hour interval between, on the 1\textsuperscript{st} October. It was assumed, the weather on this day was sunny and the sky was clear. The change of solar radiation intensity during the day was plotted in Figure 7.4 a. The HST was oriented to the west. Figure 7.4 b illustrates the solar ray
direction angles at 11:00 on the day. It was clear that the south-side passengers perceived more solar load.

![Image of graph showing direct and diffuse solar radiation]

\[ \text{Direct solar radiation} \]
\[ \text{Diffuse solar radiation} \]

**Figure 7.4** Solar radiation characters (a. change of solar radiation intensity during the day; b. direction of solar radiation)

The emissivity rate of all the interior surfaces including passengers, cabin walls, roof and floor was set up to 0.95 (Lee et al., 2014). Considering the visual effect, mechanical strength and thermal conductivity, HST cabin windows are usually composed of high-quality double-layer glass (Bouvard et al., 2018). The window
internal emissivity rate was 0.88. As a semi-transparent media, the transmittance of window was 0.2 (MRPRC, 1987).

In realistic operations, roller curtains as shown in Figure 7.5 are usually used to avoid passengers being directly exposed to solar rays. Therefore, a white roller curtain was also added to investigate the effects of applying curtain on thermal comfort. The shading coefficient of the curtain was 0.5 according to Standard TB 1951-87 (MRPRC, 1987). The overall transmissivity of the cabin window which consists of the double-glazed window and roller curtain was reduced into 0.1.

![Figure 7.5 A typical roller curtain used in HST cabin](image)

**7.2.4 Thermal Comfort Index Analysis**

According to ASHRAE Standard 55 (ASHRAE, 2004), human thermal comfort is the condition of the mind that expresses one’s satisfaction with the thermal environment. To achieve such condition, the metabolic heat generated from a human body should equal to the heat loss from the body. It means a constant heat exchange between a human body and the environment should be maintained. Therefore, an important objective of the ventilation system in an HST cabin is to create a comfortable thermal environment.

For each passenger, comfort variables include air velocity, air temperature, radiant temperature, relative humidity, and turbulence intensity in the occupied zone (ASHRAE, 2004). In order to quantify the ventilation performance, the Predicted Mean
Vote (PMV), which is accepted as the most recognized criteria index in evaluating the combination of thermal comfort variables (Lin et al., 2005), was calculated based on the Fanger’s heat balance equation (ASHRAE, 2004).

\[
PMV = [0.303 \exp(-0.036M) + 0.028] \times ((M - W) \\
- 3.05 \times 10^{-5} \times [5733 - 6.99(M - W) - P_a] \\
- 0.42 \times [(m - w) - 58.15] - 1.7 \times 10^{-5} \times M \times (5867 - P_a) \\
- 0.0014M \times (34 - T) \\
- 3.96 \times 10^{-8} \times f_{cl} \times [(T_{cl} + 273)^4 - (T_r + 273)^4] \\
+ f_{cl} \times h_c \times (T_{cl} - T))
\]

The clothing surface temperature \( T_{cl} \) in the above equation is determined by

\[
T_{cl} = 35.7 - 0.028(M - W) \\
- I_{cl}(3.96 \times 10^{-8} \times f_{cl} \\
\times [(T_{cl} + 273)^4 - (T_r + 273)^4 + f_{cl} \times h_c \times (T_{cl} - T)])
\]

The convective heat transfer coefficient \( h_c \) is determined by

\[
h_c = 2.38(T_{cl} - T)^{0.25} \text{ for } 2.38(T_{cl} - T)^{0.25} \geq 12.1u^{0.5}
\]

\[
h_c = 12.1u^{0.5} \text{ for } 2.38(T_{cl} - T)^{0.25} < 12.1u^{0.5}
\]

The clothing surface area factor \( f_{cl} \) is determined by

\[
f_{cl} = 1.05 + 0.645 \times I_{cl} \text{ for } I_{cl} \geq 0.078
\]
\[ f_{cl} = 1.00 + 1.290 \times I_{cl} \text{ for } I_{cl} < 0.078 \]  

Due to the fact that there is a large variation of people’s perception on ambient thermal comfort, an evaluation criterion is setup according to the ISO 7730 (ISO, 2005). The criterion specifies three classes of satisfaction: 1) Class A: PMV ranged from -0.2 to 0.2 represents the highest satisfaction to the environment; 2) Class B: \(-0.5 < \text{PMV} < +0.5\) is the moderate satisfaction level; 3) Class C: \(-0.7 < \text{PMV} < +0.7\) is minimum requirement for thermal comfort.

Based on the value of PMV, the prediction of the percentage of dissatisfaction (PPD) of the thermal comfort was calculated:

\[ PPD = 100 - 95 \exp(-0.03353PMV^4 - 0.2179PMV^2) \]  

However, the influence of turbulence intensity is included in the formula of neither PMV nor PPD but the turbulence of airflow has a significant impact on the thermal comfort. Thus, the percentage of dissatisfied (PD) people due to draft model was also applied for the thermal comfort analysis, expressed as follows:

\[ PD = (34 - T)(U - 0.05)^{0.62}(3.14 + 0.37u \times Tu) \% \]  

\[ Tu = 100 \times (2k)^{0.5}/u \% \]

According to the stipulation of ISO 7730 (ISO, 2005), the maximum suggested PPD value is 10%, while the maximum suggested PD value is 15%. For calculating the PMV, PPD and PD, the above equations were implemented on top of the normal CFD program executions.
7.3 Results and Discussion

7.3.1 Validation of Numerical Model

For the HST cabin ventilation model, it has been validated in author’s previous publications. Here, the radiation model was validated using Rodler’s experimental data (Rodler et al., 2015) and Gendelis’s simulation results (Gendelis and Jakovics, 2008).

Rodler’s experiment was carried out in a thermal test building at the BESTLab (Building Envelop and Solar Technology laboratory), which is located in 48 22’N 249’E. The model represents a single room with 2.97m in depth, 2.89m in width and 2.82m in height. A double-glazed window with 1.317m height and 1.317m width was installed in the west wall. A total of 23 air temperature sensors were fitted around the rooms to capture the experimental conditions. An infrared camera was installed to measure the surface temperature distribution on the floor at a specifically time. Figure 7.6 a illustrates the light intensity on the floor and surroundings. A corresponding validation computational model was set up based on Rodler’s experimental conditions and simulated using above solar radiation models. Figure 7.6 b presents the simulation results of radiative heat flux intense on the floor surfaces. It can clearly present the features of light intensity on the floor from the experiment. The temperature field of experiment result and simulation result on the floor are compared in Figure 7.6 c and Figure 7.6 d. Overall these two figures have a very good match. However, the temperature in the area on the west-side of floor was 1 to 3°C higher in the experiment. This is because the simulation was based on steady-state while the experiment was a continuous process. Thus, with the sun direction change along the time, the previous heat residue on the floor was still captured by the infrared camera.
In addition to the validation of the solar model with experimental results, this model was also quantitatively validated using the Gendelis’s numerical result. In Gendelis’s model, a living-room with different boundary construction and appropriate heat consumption was set up in the winter condition. The radiation heat transferred from a heater’s surface and through window from outside sunbeams. Gendelis’s simulations were based on Monte Carlo model in ANSYS CFX. A corresponding validation computational model was set up based on Gendelis’s model. Figure 7.7 a shows the temperature on the floor of the living-room. It is obvious that for both cases, the areas underneath heater and exposure under sunshine have significant heater temperate due to
radiation efforts. Figure 7.7 b illustrates the comparison results of velocity field in the cross-section plane of the living-room between Gendelis’s results (black vectors) and the validation simulation results (colorful vectors). In the left side plane, a very good match could be found in both cases that two strong up-ward streams existing above the heater and the floor surface which is exposure under sunshine. These two streams were the thermal plumes induced by the higher surface temperature regions as shown in Figure 7.7 a. However, there is a discrepancy in the area near the right-side wall. This could be because of uncovered factors such as wall material setup difference. Figure 7.7 c. also presents the velocity from both validation and Gendelis’s result on the red line in Figure 7.7 b. This line located above the sunshine heated floor. Both results show the similar changing pattern that velocity increased near heated floor, decreased in the middle of the room, and relatively larger velocity existing near the roof.
In general, the simulation results show good agreement with Rodler’s experimental result and Gendelis’s numerical result, and the mathematical model and numerical procedures have been proved to be acceptable for simulating the HST cabin radiation condition.
7.3.2 Comparison of Simulation Results with and without Solar-radiation

The ventilation performance and thermal comfort characteristics of HST cabin interior environment were studied under two conditions respectively, naming ventilation with and without solar-radiation. For the no solar-radiation case, the ventilation scheme was setup based on the thermal load calculation as mentioned above. In the other case, the solar radiation at 14:00 was added into the simulation. The results of two simulation scenario were analysed in terms of ventilation performance, the absorption of solar heat flux, the change of cabin air temperate and thermal comfort index.

![Comparison of temperature and velocity in the Plane Z1](image)

*Figure 7.8. The comparison of temperature and velocity in the Plane Z1 (a. without solar-radiation; b. with solar-radiation at 14:00; c. The temperature profile on one typical line)*

When comparing the ventilation performance, one plane named as Plane Z1, which crossing through the middle of the eighth-row passengers (see Figure 7.1), was chosen. The velocity vector and temperature contour on the Plane Z1 were shown in Figure 7.8.
Two cases had similar overall airflow pattern that two streams of fresh air were combined in the middle of the cabin roof area and travelled downward to the passengers sitting near the aisle. Comparing the simulation results with HVAC standards, both cases showed acceptable ventilation performance that the maximum air speed in the cabin was less than 1 m/s and the temperature was 299-303k. However, ventilation performance varied in some local areas. The air temperature around passengers in Figure 7.8 b was slightly higher than the temperature in Figure 7.8 a. Thus, more significant body thermal plumes were observed above the passengers in Figure 7.8 b. It led to a relatively strong eddy current airflow existed above two left passengers. Such eddy airflow was harmful to exhaust contaminants in this region. Also, the downward cool flow velocity near the aisle area was reduced due the effect of the strong passengers’ heat flux. It can be seen in Figure 7.8 a. that temperatures near both side windows were same. But it is clear in Figure 7.8 b that the temperature near the right-side window was much higher than the left side. This phenomenon was caused by the solar ray direction that more solar ray transmitted through the right-side window. The average temperature shown in Figure 7.8 a. was 299.54K and in Figure 7.8 b was 300.60K. Thus, the average temperature in the HST cabin was increased by 1.06K due to the solar radiation effect. Based on Moo’s results, there could be a 1-2k temperature increase after applying the solar radiation in a typical car simulation model (Moon et al., 2016). The similar phenomenon exists in a typical HST cabin that cabin temperature increases after considering the solar ray in the simulation model.

To quantitatively analyses the effects of solar radiation, a vertical line in front of the passenger sitting next to a south-side window was chosen (as shown in Figure 7.8 c.). The temperature profiles on this vertical line from two simulation results were compared. In Figure 7.8 c., the solid line refers to the no-radiation scenario, while the dashed line represents the scenario which considers the solar radiation. At the vertical 1.3-meter position, a relatively small temperature could be observed in both cases because this area was located next to a fresh air inlet diffuser. In the 0.4-1.2m region which is next to a window, the temperature was uniform for both cases. But it was clear the temperate was 1.5K higher in the case with solar radiation. In the region between 0-0.4m in height, such difference increased to 3.5K. The airflow in this area suffered not
only the transmitted solar radiation from windows but also the reflected energy from passengers. Thus, more significant increase of the airflow temperature could be observed in the near-passenger fields.

![Solar Heat Flux Contour](image)

**Figure 7.9.** The solar heat flux on the surfaces (a. The absorbed solar heat flux by cabin interior walls; b. The absorbed solar heat flux by manikins and seats)

In order to illustrate how the solar radiation affected the HST cabin, the solar heat flux contour was plotted in Figure 7.9. Based on the calculation of Solar Calculator in Fluent (ANSYS, 2016), at 14:00 in Shanghai, the direct solar irradiation at earth surface was 905.269 W/m² and the direction vector of solar ray was X: 0.56, Y: 0.68, Z: -0.48. Thus, the solar ray came into the cabin from the south-side windows. Figure 7.9 a, illustrate the intense of solar heat flux on the interior-side surfaces of cabin walls and windows. Because the material of cabin wall, roof and floor were opaque, solar ray could not transmit through these boundaries. Thus, the value of solar heat flux on the interior-side surfaces of cabin walls was extremely small. The double-glazed windows
were semi-transparent when setup the boundary type. Hence, the value of solar heat flux on the south-side windows was around 61.4 W/m². This value presented the energy of solar radiation that was absorbed and transmitted by the double-glazed windows. As above-mentioned, the absorptivity of window was 0.1 and the transmittance was 0.2. Most of solar radiation energy was reflected into the ambient. Except from the solar energy absorbed by the window and reflected into the ambient, the rest solar energy was transmitted into the cabin. Figure 7.9 b, presents the seat and passenger energy absorption of the incoming solar energy. It can be seen that only the seats and passengers next to the south-side windows suffered a high solar heat flux of 60-120W/m². Compared with 905.269 W/m² total solar irradiation on earth surface, the solar heat flux on passengers seated under the solar ray was much smaller.

![Figure 7.10](image)

Comparing the solar-radiation effects on thermal comfort, the distribution of PMV value of each passenger was shown in Figure 7.10. Generally, most the PMV values calculated based on both simulation cases (with and without solar radiation) indicated the internal thermal environment could meet the minimum thermal comfort requirement. In the no-radiation case, most areas were considered as cold regions where PMV values were smaller than 0. Particularly, the PMV value in the aisle area was around -0.59 due to the strong downward supply air flow. A prominent increase of
PMV value was seen in the case with solar radiation. In the aisle area, the coldest region, the PMV value increased to -0.31 in this case. The PMV value was much higher in the region near the south-side windows than the rest areas, which is induced by the solar radiation.

Figure 7.11. The distribution of PPD index (a. Without solar-radiation; b. With solar-radiation at 14:00)

In the passenger activity plane (Plane Y1, see Figure 7.1), the distribution of PPD values were also plotted in Figure 7.11. It was clear that without solar-radiation, the changing gradient of PPD value was larger. Similar as the PMV distribution that the coldest area mainly concentrated in the aisle, the most thermal dissatisfactory area was also the aisle where the worst PPD value was 25%. However, PPD values were much smaller in other areas. In this plane, the average PPD value was 9.76%. After adding the solar radiation into the simulation, the average cabin internal temperature increased and the velocity magnitude in the aisle area decreased. The highly dissatisfactory area was draw back to 19% in the aisle area, and the distribution of PPD value in Plane Y1 was more uniform. The average PPD value was reduced to 7.86% after applying the solar radiation.

7.3.3 The Solar-Radiation Effects at Different Time in One Day

As above-mentioned, the solar-radiation was time-dependent because the incident angle and solar load changed with time. Considering the incident angle, for the location of Shanghai, the sun rises in southeast and set in the southwest. Solar-load on the earth
surface increased to the maximum at noon time and gradually reduced in the afternoon. To investigate the influence of solar-radiation at different times, temperature contours on two typical passengers were firstly studied. Additionally, thermal comfort indexes, including PMV, PPD and PD at different times were analysed.

Figure 7.12. Surface temperature of two passengers seating next to the window

The surface temperature contours on two passengers (P-A, P-B) seated next to a south-side window (as shown in Figure 7.12) were plotted with different time stage setups. Because the HST was assumed to travel to the east, these two passengers suffered the
sun-shine at their backs in the morning. The sunshine was partially blocked by seat
backs. Therefore, the surface temperature on left and right-side shoulders of P-A at
10:00 were similar. In the later stages, the surface temperature of P-A gradually
increased. At 12:00, most surface area of P-A suffered a high temperature. Before
12:00, the most obvious temperature change area was on the legs of P-A. After 12:00,
the strongest temperature effected area was on the body of P-A. Due to the effects of
strong solar-load and human body heat flux, the largest area of high temperature
surface was observed at 13:00 on P-A. At 14:00, the temperature difference between
north-side and south-side shoulders of P-A was extremely obvious. After 14:00, the
overall surface temperature of P-A largely reduced. However, at the 16:00, the surface
temperature of P-A was quite uniform, and the average surface temperature of P-A was
slightly higher than it of P-B. This was because the sunshine was mainly blocked by the
cabin wall at this time. Similar as the surface temperate change of P-A, the surface
temperature of the passenger seat also increased in the morning and then decreased in
the afternoon. But the change of surface temperature of P-B, who was positioned next
to the aisle, could barely be noticed.

To investigate the change of thermal comfort index along with the change of time, the
average manikin surface temperature (Figure 7.13 a), average cabin air temperature
(Figure 7.13 b), average PMV (Figure 7.13 c), average PPD (Figure 7.13 d), and
average PD (Figure 7.13 e) were plotted. It can be seen from Figure 7.13 a and b that
the average manikin surface temperature and average cabin air temperature had similar
changing trends during the day. From morning till noon, the cabin interior temperature
and manikin surface temperature both increased. Both temperatures reached their
maximums at 14:00 then decreased quickly. But the range of temperature change of
average manikin surface temperature was larger than it of the cabin air temperature. It
is because the passengers seated next to a south-side window were directly exposed to
the solar ray, and they were more sensitive to solar radiation. The average PMV and
PPD values slightly varied with time. Figure 7.13 c illustrates that passengers could feel
a little bit cold (average PMV: -0.15) in the morning and late afternoon. But it was a
little hot (average PMV: +0.12) in the cabin at noon. The average PPD changed from
6.3% to maximum level of 6.9% in the late afternoon, which was still smaller than the
maximum suggested PPD level of 10%. The average PD value was between 11-13% in the HST cabin as shown in Figure 7.13 e. This range is within the ISO 7730 (ISO, 2005) comfort standard of 15%. Even though, the average cabin thermal comfort indexes were within the thermal comfort standards, for the passengers exposed to the solar ray, these indexes were out of the standard ranges, and it would be explained in Section 3.4.

![Figure 7.13 Thermal comfort index change with time profiles](image)

7.3.4 The Solar-Radiation Effects with Applying the Roller Curtain

Based on previous analysis, surface temperature of the passengers exposed to the solar ray was much higher than it of the normal human skin temperature. As mentioned, window roller curtains would usually be laid down. Thus, it was necessary to investigate the thermal comfort in the HST when a curtain was used. The simulation included the use of white roller curtains on all windows at 14:00. Simulation results were compared with the case where no curtain was used but all the rest boundary conditions were same. One plane (Plane X1, see Figure 7.1) crossing through the
middle of passengers sitting next to a south-side window was chosen. Temperature and PMV value on this plane were shown in Figure 7.14. The thermal comfort indexes for the passenger exposed to the solar ray were presented in Table 7.2.

![Figure 7.14. Comparisons of temperate and PMV contour for passengers exposed to the solar ray](image)

It was clear that the average cabin temperature was effectively reduced by 0.75 K after using the curtain. Also, the temperature gradient in the cabin was small. This was a more uniform temperature environment. The reduced temperature largely improved the thermal comfort in this domain. Before using the curtain, most cabin areas had higher temperature profiles than the comfort criteria except the regions near inlet diffusers. After adding the curtain, the PMV values in most regions were reduced to +0.1 which was ideal for thermal comfort.

A more obvious improvement was found on passengers’ surface areas. Passengers who were prolonged exposed to the sun had their surface temperatures increased to 320K which was an unhealthy temperature level for humans. But the intensity of the solar ray was largely reduced after transmitting through a double-glazed window and then a curtain. Thus, the passenger surface temperature dropped to 310K. The similar change of thermal comfort indexes could be observed (see Table 7.2). As these passengers
were exposed to the sun, all the PMV, PPD and PD values were out of standard comfort ranges, which should be less than +0.5, 10% and 15% respectively. After using the curtain, the thermal comfort indexes changed into comfort ranges, with PMV, PPD and PD values improved by 25.4%, 39.6% and 33.1% respectively.

Table 7.2 Comparison of thermal comfort index for passenger exposed to the solar ray

<table>
<thead>
<tr>
<th>Thermal comfort index</th>
<th>PMV</th>
<th>PPD (%)</th>
<th>PD (%)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Double glazed window</td>
<td>+0.59</td>
<td>12.13</td>
<td>18.14</td>
</tr>
<tr>
<td>Double glazed window with curtain</td>
<td>+0.15</td>
<td>8.69</td>
<td>12.14</td>
</tr>
</tbody>
</table>

| Improvement            | 25.4% | 39.6% | 33.1% |

In conclusion, the roller curtain could effectively reduce cabin average temperature. Especially for the passengers exposed to the solar ray, a large reduction of surface temperature by 10K could be achieved, meanwhile the thermal comfort could be improved. Using a window curtain in hot weather instead of adjusting the internal temperature controller could well maintain the ride comfort and to reduce energy consumption.

7.4 Conclusions

The presented study numerically investigated the combined heat transfer characteristics of solar radiation and evaluated the thermal comfort in a HST cabin under such condition. The following conclusions were made:
● The overall cabin air temperature, especially the temperature near passengers, was significantly increased after applying the solar radiation effects in the simulation. Such phenomenon resulted in an increase of average PMV value. The distribution of PD value in HST cabin come to be more uniform with the highest PD value changed from 25% to 19%.

● Passengers sitting next to the south-side windows had an obvious surface temperature change at different time of a day, especially in the leg and shoulder areas. The average cabin temperature increased in the morning and decreased in the afternoon. Similar change was seen on the average PMV values with a variate of 0.27. The average PPD and PD values also changed with time among 6.3-6.9% and 11-13% respectively. While the overall average thermal comfort indexes were under thermal comfort standard, these indexes were out of comfort standard for the passengers exposed to the solar ray.

● A roller curtain could effectively reduce the cabin average temperature. Particularly, for passengers exposed to the solar ray, the surface temperature could be reduced by 10K with reasonable thermal comfort indexes achieved. To improve the ride comfort and reduce energy consumption in hot weather, using the curtain could be a good option.
Chapter 8
Multi-objective Optimization of Air Quality in HSR Cabins

The main findings of this chapter have been included in:


With growing of world population, the energy problem is gathering more and more attention from the society. Heating, ventilation and air conditioning (HVAC) system engineers are facing challenge of maintaining high level of thermal comfort and indoor air quality for occupants while minimizing the system energy consumption. Traditional way of handling multi-objective optimization problem aggregates each design objective into one single objective function through biased weights, which usually consumes massive time to find proper solutions as the results are sensitive to the chosen values of the weights. This paper proposed a multi-objective optimization platform using nondominated sorting-based particle swarm optimization (NSPSO) algorithm to perform multi-objective optimal design of the ventilation system in a fully occupied high-speed train (HST) cabin. A computational model of HST cabin occupied by four
rows of passengers was created in ANSYS Fluent where high resolution computational thermal manikins were used to increase the simulation accuracy. Different combinations of ventilation operation parameters were investigated against its performance in terms of thermal comfort, air quality and energy consumption. A Multi-fidelity Kriging surrogate procedure is also proposed to reduce computational cost by replacing part of fine mesh CFD simulations with coarse mesh ones while maintaining an acceptable accuracy. This paper demonstrates that the presented approach can deal with multi-objective optimization in indoor ventilation system design more efficient with up to 35.61% reducing of computational time.

8.1 Introduction

High-speed trains (HST) has emerged as an alternative fast public transportation around the world due to their huge transport capacity and high efficiency (Yang et al., 2017). Unlike other public transportation vehicles (e.g. buses, normal trains), an HST cabin is an isolated space with high occupant density where passengers are normally staying inside for couple hours throughout the journey. The heating, ventilation and air conditioning (HVAC) systems is therefore the key of controlling the cabin environment which has a direct impact on the passengers’ thermal comfort and health. On the other hand, while providing a comfort environment for passengers, air conditioning system consumes around 75% of all other non-propulsive energy consumption in the cabin (Liu et al., 2015). The investigation of optimal design of the ventilation system in HST cabin to ensure passengers’ thermal comfort and health while minimizing energy consumption has become of significant importance to the railway industry.

To obtain the optimal design, in the conventional “trial and error” design cycle, system design parameters such as supply airflow rate, air temperature and humidity are manually adjusted and evaluated based on on-site measurements or analytical and empirical models (Liu et al., 2015). Over the past decades, with the rapid development of computer technology, computational fluid dynamics (CFD) techniques have been widely adopted to predict the indoor air environment aiming to shorten the time and reduce the cost of the lengthy HVAC system design cycles in buildings (Ravikumar
and Prakash, 2009, Cardinale et al., 2010, Hiyama et al., 2010, Kochetov et al., 2015, Limane et al., 2018, Tian et al., 2018, Cao et al., 2014, Pu et al., 2014, Wang and Zhai, 2016, Yan et al., 2016) and long-haul transportation cabins (Liu et al., 2012a, Li et al., 2016b, Liu et al., 2013, Kwon et al., 2009, Zhang and Li, 2012, Konstantinov and Wagner, 2014, Konstantinov and Wagner, 2015). With no doubt, comparing to the on-site measurement or experimental analysis, the CFD technique appears to be a cost and time effective tool for design optimization. Nevertheless, due to the nature of trial and error design process, a large number of simulations is usually required for covering the entire design space; leading to a significant computational time and resource.

In order to reduce the computational time, some surrogate techniques such as artificial neural network (ANN) (Zhou and Haghighat, 2009a, Zhou and Haghighat, 2009b, Acikgoz et al., 2017, Bre et al., 2018), Support Vector Machine (SVM) (Mousa et al., 2017), Kriging (Li et al., 2017) and Proper Orthogonal Decomposition (POD) (Li et al., 2013) are employed as a fast alternative approach replacing the CFD simulation to approximate the nonlinear and complex behaviour of the indoor airflow. On the other hand, aiming to automate the trial and error process, evolutionary algorithms (EA) have been proposed and coupled with the CFD to search the globally optimal solution (Luh and Lin, 2011, Li et al., 2013, Zhai et al., 2014). Although the CFD-EA coupled approach significantly reduces the required number of CFD simulations to reach optimal solution, it still requires a substantial amount of CFD simulations for training the surrogate models to construct a reliable response space for EA.

Furthermore, the evaluation of indoor environment can be a complex process where multiple evaluation criteria are normally involved such as air velocity and temperature near the occupants, contaminant concentrations, percentage dissatisfied of draft, age of air, total energy consumption, etc. The system design is therefore a multi-objective optimization process where trade-off relations among those design indices are usually needed to be considered. Especially, in terms of indoor thermal comfort evaluation, lots of research works have been done by Ricciardi’s group (Buratti and Ricciardi, 2009, Buratti et al., 2013, Nematchoua et al., 2014, Ricciardi and Buratti, 2015, Buratti et al., 2016, Ricciardi et al., 2016). In most of the previous works, in order to handle multi-
objective problem, all the design objectives are aggregated into one single objective function through pre-defined biased weighting factors (Laverge and Janssens, 2013, Li et al., 2013). One of the major disadvantages of this method is that the optimal design could be sensitive to the weighting factors, thus different values of the weights could result in substantially different solutions. Therefore, the values of these weighting factors are object to professional knowledge and expert judgements. In addition, the optimization procedure provides only one optimal solution per simulation run, giving the designers no flexibility in selecting alternative solutions for striking a trade-off of the conflicting parameters.

In attempting to address the above shortcomings, we propose a novel design scheme where a nondominated sorting-based particle swarm optimization (NSPSO) algorithm is utilized to achieve multi-objective optimization without using any biased weights. Furthermore, in order to minimize the computational cost in the design process while without sacrificing the accuracy, an improved surrogate method - Multi-fidelity Kriging algorithm is established and adopted in the present study. This improved Kriging algorithm is capable to accurately capture the system characteristics extract from CFD simulations yet requires very limited computational time for constructing the training database. The proposed design approach is tested and verified on a full-size computational HST model which is assumed to be fully occupied by passengers. The rest part of this paper will describe the details of the design procedure and discussions of the validation results.

8.2 Methods

8.2.1 Particle Swarm Optimization (PSO) and Nondominated Sorting-based PSO

As one of the evolutionary algorithms (EA), the particle swarm optimization (PSO) has been widely adopted in solving many engineering optimization problems, especially for those non-convex and multimodal engineering problems where traditional gradient-based mathematical programming methods have difficulties in identifying the optimal
solution (Deb, 2001). Furthermore, previous study has also proved that the PSO has a faster convergence rate in optimization in comparison to other population-based stochastic optimization methods (e.g. genetic algorithms) (Hassan et al., 2005). As a brief history, the PSO was firstly introduced by James Kennedy based on the inspiration drawn from observations of the social behaviours of insects including learning from previous experience and communicating with successful individuals (Kennedy, 2001). In the PSO, each particle has its own position and velocity, which are represented by \(x_i\) and \(v_i\), respectively. The position and velocity of the particle are updated according to the following equations:

\[
\begin{align*}
    v_i(t+1) &= \omega v_i(t) + c_1 \varphi_1 (p_i - x_i(t)) + c_2 \varphi_2 (p_g - x_i(t)) \\
    x_i(t+1) &= x_i(t) + v_i(t+1)
\end{align*}
\]

where \(p_i\) and \(p_g\) represent the personal best position and global best position, respectively, and the \(c_1\) and \(c_2\) are two uniform random numbers within the range \([0, 1]\). The \(\varphi_1\) and \(\varphi_2\) are two constants which are usually set to 2. The parameter \(\omega\) decreases with iterations within the range \([0.4, 1.2]\). To avoid going out of the search space, both the position and velocity are limited within boundaries, \([x_{min}, x_{max}]\) and \([v_{min}, v_{max}]\), respectively. Nevertheless, it is worth noting that the original PSO can only provide solutions for single-objective optimization problems.

Inspired by the works done by Deb (Deb et al., 2002), Li proposed a Nondominated Sorting Method to extend the original PSO to multi-objective optimization problems (MOP) - namely Nondominated Sorting based Particle Swarm Optimization (NSPSO) (Li, 2003). In the NSPSO, the updating equations for particle position and velocity remain unchanged, while the selection of the personal best and global best has been re-designed. Two main mechanisms are used to determine the global best among the population – 1) nondominated sorting for identifying different fronts, and 2) crowding distance computed for particles within each front to encourage solution diversity. These kinds of information are used to select suitable leaders (i.e. global best) at each iteration to guide the particles moving towards the Pareto-optimal Front while still maintaining a good distribution of solutions along the Pareto-front.
8.2.1.1 Nondominated Sorting

Figure 8.1 shows an example of the nondominated sorting process. Considering 2 objectives (i.e. $f1$ and $f2$) to be optimized in the process, the entire population (i.e. the 10 particles that labelled as 1 to 10) is sorted into different levels of fronts according to the domination comparisons between particles. The particles in same front are nondominated with each other. As depicted in Figure 8.1, Front 1 is the highest-level nondominated front because all particles in it are not dominated by any other particles in the entire population. The main goal of nondominated sorting is to classify the whole population into different levels of nondominated fronts. The global best particle (leader of the population) can be randomly selected from the highest-level front. This kind of selection process will push the whole population towards the true Pareto Front. More information regarding nondominated sorting can be found in (Li, 2003).

![Figure 8.1 An example of the nondominated sorting process in NSPSO.](image)

8.2.1.2 Crowding Distance

Different from a single-objective optimization, maintaining the diversity in a set of solutions is vital in a multi-objective optimization (Deb et al., 2002, Li, 2003). Throughout the optimization process, the leader must be selected properly to avoid local optimal aggregation of the whole population. In NSPSO, computing the crowding distance values among particles in the highest-level of nondominated front is used to
select leaders that are both good and far apart from each other. Inspired by (Deb et al., 2002), we introduced a new way to calculate the crowding distance. Figure 8.2 shows an example of the crowding distance among particles. For each particle, the crowding distance is defined as the following:

\[
D_n = \begin{cases} 
\infty & n = 1, N \\
\sum_{i=1}^{n} d_{n-1} + d_n & 1 < n < N 
\end{cases}
\]

8.2

The particle with a higher crowding distance value will have a high probability to be selected as the leader. Consequently, particles in the top front are likely to maintain a good level of population diversity.

![Figure 8.2 Crowding distances among individuals in the highest-level nondominated front.](image)

**8.2.2 Kriging and Multi-fidelity Kriging**

As demonstrated in previous studies (Liu et al., 2013, Tian et al., 2018), it is certainly possible to use numerical model predicting the airflow behaviours inside buildings or vehicle cabins. However, as discussed above, the nature of CFD simulations is inherently time consuming and computationally exhaustive, making it impractical for design optimization. Surrogate techniques (also referred as Meta methods in literatures) have been adopted as a fast alternative to the CFD simulations, such as ANN (Zhou and Haghighat, 2009a, Zhou and Haghighat, 2009b) and SVM (Mousa et al., 2017).
Nonetheless, to achieve a reliable and accurate prediction, both ANN and SVM require large amounts of training data generated from numerical simulations. The construction of training database remains a significant computational burden jeopardising the progress of the optimization process. Alternatively, the Kriging method has aroused much attention due to its capability in achieving high prediction accuracy with relatively small training sample size (Li et al., 2017). Detailed formulations of Kriging can be found in literature and the references therein (Handcock and Stein, 1993, Van Beers and Kleijnen, 2004, Sivia and Skilling, 2006, Wikle and Berliner, 2007, Forrester et al., 2008). For the ease of the reader, a brief introduction to Kriging is presented as following.

Kriging was originally developed for geostatistical application by Danie G. Krige (Krige, 1951) to estimate the most likelihood of gold distribution based on a small size of samples from a few boreholes. It can also be adopted to predict the Gaussian process governed by prior covariance in other engineering fields (Kleijnen, 2009). The basic idea of Kriging is to predict the value of a function at a given point by computing a weighted average of the known values of the function in the neighbourhood of the point, which is expressed as:

$$\hat{z}_o = \sum_{i=1}^{n} \lambda_i z_i$$  \hspace{1cm} \text{(8.3)}

where \(\hat{z}_o\) represents a local estimation at the data location \((x_o, y_o)\), \(z_i\) is the known sampled value at the data location \((x_i, y_i)\) and \(\lambda_i\) represents the weighting coefficient which can be calculated by minimizing the estimation variance:

$$\min_{\lambda} Var(\hat{z}_o - z_o)$$  \hspace{1cm} \text{(8.4)}

subjects to the unbiased condition:

$$E(z_o - \hat{z}_o) = 0$$  \hspace{1cm} \text{(8.5)}

and the normalization condition:
The weighting coefficient $\lambda_i$ in Eq.8.1 can be solved using a quasi-Newton optimization method or other similar algorithm (Gano et al., 2006). After obtaining the weighting coefficient, the predicted value of the unobserved location can be evaluated based on Eq.8.1.

Benefit from the formulation, Kriging poses a great saving of computational resources without going through the tedious simulation process as required in CFD technique. Similar to other methods, Kriging requires training data extracted from CFD simulation results. In terms of CFD simulation, it is well known that high mesh resolution is crucial to minimize the discretization and truncation errors in the numerical procedures. The high mesh resolution imposes significant computational time and cost in constructing the training data. To remedy this problem yet retaining the same level of prediction accuracy, we adopt a Multi-fidelity Kriging method where the training data could be extracted from a small sample of high-quality mesh while majority of the training samples are constructed by computationally efficient low resolution mesh.

Multi-fidelity Kriging, also known as Multivariate Kriging or Co-Kriging, was originally developed for mineral explorations where measurements of different ores are available (de Baar et al., 2015). It estimates or predicts for a poorly sampled variable (low-fidelity data) with help of a well-sampled variable (high-fidelity data). A full derivation of Multi-fidelity Kriging can be found in (Forrester et al., 2007) and the reference therein. For example, we assume that we have two sets of data yield to Gaussian process:

$$\sum_{i=1}^{n} \lambda_i = 1$$
\[ Z_1(s) = \mu_1 + \varepsilon_1(s) \]
\[ Z_2(s) = \mu_2 + \varepsilon_2(s) \]

where \( Z_1 \) represents the high-fidelity process and \( Z_2 \) represents the low-fidelity process, \( \varepsilon_1 \) and \( \varepsilon_2 \) are the random errors contained in the high-fidelity process and low-fidelity process, respectively. There exists autocorrelation for each process and cross-correlation between them. Multi-fidelity Kriging attempts to predict the high-fidelity process \( Z_1(s_0) \) using information (autocorrelation and cross-correlation) in the covariate \( Z_1 \) to make a better prediction, as the following:

\[ Z_1 = \rho Z_2 + Z_D \]

where \( \rho \) is regression coefficient and \( Z_D \) is a new process representing the difference between the high-fidelity process and the low-fidelity process, which will correct the low-fidelity data.

### 8.2.3 Workflow of the Design Procedure

With the NSPSO and Multi-fidelity Kriging, an efficient design optimization process is introduced. The overview of the proposed optimization procedure is shown in Figure 8.3. As depicted, at the beginning of the process, the user will determine the range of the design variables as input and the corresponding sampling locations within the design space. Afterwards, the user will construct the computational model for the specific design case. In the meshing of the computational model, a mesh independent analysis is required to decide the proper grid size for the fine mesh case and coarse mesh case. Next, multiple simulations will be carried out with high and low mesh resolution for constructing the training data for the Multi-fidelity Kriging. Following the training based on the CFD simulated results, the multi-objective optimization NSPSO algorithm will be adopted for searching the global optimal solutions (Pareto Front), in which Multi-fidelity Kriging will be used for evaluating the objective values.
of each particle. The details of the CFD model and optimization results will be discussed in later sections.

![Diagram of the CFD-based multi-objective optimization approach with Multi-fidelity Kriging](image)

**Figure 8.3** Framework of the CFD-based multi-objective optimization approach with Multi-fidelity Kriging

### 8.3 Computational Model Description

#### 8.3.1 Model Preparation and Validation

In this study, the CRH2 model – a typical HST model in China has been chosen as the ventilation design environment (Wang et al., 2014). The CRH2 model cabin has the 3-2 seat arrangement (Figure 8.4). In order to save computational resources, instead of using a full HST cabin, four rows cabin section was built as the computational domain with the front and back plane being specified as translational periodic boundaries. The approximate periodicity of air flow field in train cabins has been verified in our previous studies (Yang et al., 2017). The HST is assumed to be operated in summer while the external ambient temperature is assumed to be 35℃. To consider the heat invasion through the cabin envelope, an equivalent heat transfer coefficient of 2.2 W/(m²K) was assumed for the material of cabin walls. The incident heat flux of solar radiation through the windows was 688.5W/m² (Yang et al., 2017). Other solid cabin walls including floor and seats were assumed to be adiabatic.
The cabin is assumed to be fully occupied by adult female passengers which are represented by the 3D-scanned manikin models. Figure 8.4 shows the detail of the computational domain. Radiative heat dissipation from the manikins surface is ignored and only convective heat transfer is considered in this study, which results in a total convective heat load of 30W (Yan et al., 2016) for each manikin. In addition, VOCs are assumed to be released from the surface of Passenger-S, who is seating in the second row of the second column (which is the second column).

Figure 8.4 Cabin geometry and detailed meshing features.

The ventilation system of the HST cabin is assumed to be operating in compliance to the prevalent railway HVAC standard – UIC 553. Air was supplied from inlet diffusers and exhausted via the return air outlets located under the seats.

The commercial CFD software – ANSYS CFX 17.0 was employed as the numerical solver. The cabin airflow field was solved using the incompressible Navier-Stokes equations, where the generic transportation equation is given by:

$$\frac{\partial \Phi}{\partial t} + \frac{\partial}{\partial x_i} \left( u_i \Phi - \Gamma \frac{\partial \Phi}{\partial x_i} \right) = S_\Phi$$  \hspace{1cm} 8.9

where $\Phi$ is the general variable; $u$ is the fluid velocity; $\Gamma$ is diffusion coefficient; $S_\Phi$ is the source term. The RNG k-epsilon turbulence model was adopted to model the air turbulence.
The above mathematic models and numerical procedures have been fully validated using a 7-row mock-up cabin. Details can be found in (Li et al., 2014). The temperature and velocity profiles along a vertical line were compared with the referred airflow measured in the mock cabin. As shown in Figure 8.5, the numerical results agree well with the experimental data.

![Figure 8.5 Comparison of temperature and velocity between experimental and simulation results.](image)

### 8.3.2 Design Objectives

In order to assess the performance of the ventilation system inside a HST cabin, the Predicted Mean Vote (PMV), Contaminant concentration, and energy consumption were adopted as the quantitative indicators to evaluate thermal comfort, air quality and energy efficiency, respectively. The definitions of these design criteria are briefly introduced as follow.

PMV (Fanger, 1972) represents the mean subjective satisfaction with the indoor thermal environment with a number between -3 (cold) and +3 (hot). The zero value is defined as the ideal representation of thermal neutrality. It is defined by:
The PMV index is evaluated based on an empirical equation which is correlated to the local air temperature, mean radiant temperature, relative humidity, air speed, metabolic rate, and clothing insulation (Fanger, 1972). In this study, we assume that the occupants are seated in quiet position (i.e. metabolic rate of 1.0 met) with a summer clothing (i.e. 0.2 clo), and we evaluated the average PMV based on the predicted field information obtained from CFD simulations.

For presenting the VOCs contaminants emitted from human skin, acetaldehyde is chosen, as it has a relative high concentration level compared with other components in VOCs. The kinematic diffusivity of acetaldehyde is $1.6 \times 10^{-5}$ m$^2$/s, and concentration measured from a normal human skin is $3.0 \times 10^{-5}$ kg/m$^3$. Due to the extremely low concentration of gaseous contaminants, it is assumed that the existence of acetaldehyde has no effects on the air flow field and the transport of VOCs was modeled using a transportable scalar:

$$\frac{\partial \varphi}{\partial t} + \frac{\partial}{\partial x_i}(U \varphi) = \frac{\partial}{\partial x_i} \left( D_{\varphi} \frac{\partial \varphi}{\partial x_i} \right) + S_{\varphi}$$

where $U$ is the fluid velocity in air-flow domain; $\varphi$ is the concentration of contaminant; $S_{\varphi}$ is a volumetric source term; $D_{\varphi}$ is the kinematic diffusivity. Similar to the average PMV, the average VOCs concentration was extracted from the predicted CFD field information.

Following the previous study (Zhou and Haghighat, 2009a, Li et al., 2013), the energy consumption of the air-conditioning system is divided into two parts: ventilation fan power and the cooling or heating load. Energy consumption in the two parts is determined as follow:
\[ E_{\text{fan}} = \frac{P \cdot V_{\text{air}}}{\eta_{\text{fan}}} \]
\[ E_{\text{cooling/heating}} = m_{\text{supply}} c_p (T_{\text{return}} - T_{\text{supply}}) \]
\[ + m_{\text{outdoor}} (h_{\text{outdoor}} - h_{\text{return}}) \]
\[ E_{\text{total}} = E_{\text{fan}} + E_{\text{cooling/heating}} \]

where \( P \) is air pressure difference of the fan and \( V \) is volume flow rate of supply air (m\(^3\) s\(^{-1}\)), \( m \) represents the mass flow rate of the air (kg s\(^{-1}\)), \( c_p \) is the specific heat capacity of air, \( T \) represents temperature, \( h \) is the specific enthalpy of air (J kg\(^{-1}\)) which is related to air temperature and relative humidity. Similarly, we can get energy costs from the CFD-Post package.

8.4 Results and Discussions

8.4.1 Design Optimization Problem for the HST Cabin Ventilation

The design optimization of this study aims to allocate the best air ventilation conditions (i.e. supply air velocity and temperature setting) for the HST cabin of where multiple objectives (i.e. thermal comfort, air quality and energy) are optimized. According to the UIC 553-1, inner temperature of the cabin should be set around 27 °C under the exterior environment condition set in this study. The supply air temperature and flow rate for optimization is evaluated based on the energy balance and the thermal load in the HST cabin. It is assumed that the thermal energy generated and absorbed inside a cabin is equal to the energy taken away by the airflow from inlet vents to outlet vents (\( Q_{\text{air}} \)). The main sources of thermal load inside the cabin are the heat flux of human body (\( Q_{\text{human}} \)) and thermal conduction of cabin wall (\( Q_{\text{wall}} \)). The energy balance in the cabin can be expressed as follow:
\[ Q_{\text{air}} = Q_{\text{wall}} + Q_{\text{human}} \]  

Based on the cabin energy balance, the fresh air supply rate at inlet diffusers is arranged from 20 to 24 m\(^3\)/h/person, the supply air temperature should be ranging from 295K to 299 K. The design ranges of the inlet temperature and mass flow rate should be 295 K to 299 K and 0.51 kg/s to 0.61 kg/s, respectively. In order to generate sufficient training samples for the Kriging prediction without significant computational burden, a total of 25 set of design parameters are firstly located and evenly distributed within the design space (see Figure 8.6). Sequentially, a total of 25 CFD simulations with different design parameters have been carried out with the identical fine mesh resolution and boundary conditions (excepts the supply air temperature and mass flow rate). All simulations were carried out in a computational domain consisting of a total of 5,951,997 nodal points. A mesh dependent study has been carried out and proven that the predicted results are mesh independent. More details are presented in the later session.

![Figure 8.6 Definitions of inlet boundary conditions in CFD simulations.](image)

After the simulations, the key air flow parameters (such as local air velocity, air temperature, contaminant concentration, etc.) were extracted from the CFD simulated result for evaluating the corresponding design objectives – PMV, contaminant concentration and energy consumption. Among the objectives, designers would like to
create the most comfortable and cleanest environment and at the same time, to minimize the energy consumption of ventilation system. In reality, it is very common that these three objectives are conflicting with each other. In order to demonstrate the trade-off relationships, we listed three groups of typical results in Table 8.1, where the first row shows the global PMV minimum point (i.e. the most thermal comfortable) while both the contaminant concentration and the energy consumption are quite large. Similarly, the following rows give other situations where contaminant concentration and the energy consumption are the global optimal, respectively. As shown in the table, all three design objectives are conflicting each other where one could only optimize a trade-off solution. The CFD simulation results of these three cases are shown in Figure 8.7, Figure 8.8, and Figure 8.9, respectively. In the figures, the left image gives the temperature distribution contour and velocity vectors on a section plane inside the cabin, and the right top indicates the PMV distributions around passengers’ body surfaces. The right bottom one demonstrates the distribution of contaminant concentration near the VOCs release source (Passenger S). By comparing Figure 8.7 and Figure 8.8, it is clearly shown that Case 1 has the better thermal comfort (i.e. Case 2 is too cold for the passengers) with the worse contaminant control. In contrast, in Case 2, contaminants are driven towards a higher elevation level of the cabin further away from breathing zone (see also in Figure 8.8). Based on the Table 8.1, Figure 8.7, Figure 8.8, and Figure 8.9, it can be observed that the design objectives are conflicting where a multi-objective optimization platform is required for obtaining the pareto-front of the ventilation design problem.

*Table 8.1 Three typical groups of values minimizing PMV, Contaminant, Energy, respectively*

<table>
<thead>
<tr>
<th>Case No.</th>
<th>T(°C)</th>
<th>V(kg/s)</th>
<th>PMV</th>
<th>Contaminant(kg/m³)</th>
<th>Energy(W)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>24.45</td>
<td>0.5731</td>
<td>-1.5E-03</td>
<td>1.52E-07</td>
<td>1646.20</td>
</tr>
<tr>
<td>2</td>
<td>22.04</td>
<td>0.5938</td>
<td>-1.2E00</td>
<td><strong>9.19E-08</strong></td>
<td>2058.44</td>
</tr>
<tr>
<td>3</td>
<td>25.85</td>
<td>0.5100</td>
<td>1.5E-01</td>
<td>1.44E-07</td>
<td><strong>1463.66</strong></td>
</tr>
</tbody>
</table>
Figure 8.7 CFD simulation results of case 1 in Table 8.1

Figure 8.8 CFD simulation results of case 2 in Table 8.1
8.4.2 Validation of the High-fidelity Kriging Method

To accelerate the optimization process, the High-fidelity Kriging algorithm is used as a fast alternative to the lengthy CFD simulations. In order to verify the accuracy of the Kriging prediction, a total of 16 additional ventilation conditions were simulated by the CFD model (i.e. the 16 green points within the design space in Figure 8.10) were carried for constructing a reliable respond surface of the objective values within the design space.
Figure 8.10 The additional CFD simulations and its corresponding ventilation conditions within the design space (i.e. green points)

The simulations conducted at red points in Figure 8.10 are used to train the Kriging algorithm. After the training, the Kriging method is used to predict the three objective values under the 16 different ventilation conditions (i.e. green points) and validated against the CFD predictions. The prediction error of the Kriging in comparison to the corresponding CFD results are shown in Figure 8.11. From the figure, one can observe that the maximum error is maintained below 5% for all design objectives. Especially, for energy prediction, the Kriging gives a prediction error below 0.6%, clearly indicating the robustness of Kriging surrogate method in capturing the response surface of the objective value with respect to different ventilation conditions.
8.4.3 Design Optimization Using NSPSO with High-fidelity Kriging

As mentioned above, a multi-objective optimization is carried out using NSPSO where the Kriging method is used for evaluating the objective values for replacing the CFD simulations. In this study, a NSPSO algorithm has been established and implemented using MATLAB R2017b. The algorithm considers two design input variables (i.e. supply air temperature and flow rate) and search the optimal design solutions based on the three objectives (i.e. PMV, energy and contaminant concentration). The size of swarm population was 100 and the maximum iteration number was 200 (this configuration balances the accuracy and computational cost). In each generation, the validated Kriging surrogate was used to calculate the fitness values for the whole particle population. The final optimal trade-off solutions (i.e. pareto front) obtained from the NSPSO within the objective space are shown in Figure 8.12. Meanwhile, the
The corresponding input distribution of the Pareto front within the design space is also illustrated in Figure 8.13.

Figure 8.12 clearly shows the trade-off relationships among all three design objectives, where Figure 8.12(a) gives a 3D Pareto front visualizing the trade-off among all the three objectives. For the ease of reading, the 2D Pareto fronts considering only two conflicting objectives are shown in Figure 8.12 b-d. In Figure 8.12, the Pareto front represents a trade-off solution which are equally good in terms of multi-objective considerations. It is worth noting that the solutions listed in Table 8.1 are only three single points as part of the Pareto front. As demonstrated, the optimization framework using NSPSO and Kriging can entirely eliminate the quest of weighting factor in objective function and obtains a set of trade-off solutions within the objective space in a fast, reliable and automated procedure. The optimized solutions allow designers to evaluate the best options of the complex problem; choosing the best design solutions based on their professional judgement and user preferences.

![Figure 8.12 Optimal solutions (‘Pareto Front’) given by NSPSO in objective space](image-url)
8.4.4 Enhancement in Optimization Process Using the Multi-Fidelity Kriging

In previous studies (Yuan et al., 1999, Lin et al., 2009, Zhou and Haghighat, 2009a, Zhou and Haghighat, 2009b, Li et al., 2012a, Li et al., 2013, Liu et al., 2015), the simplified manikin models such as rectangle or cylinder blocks are used to represent occupants or residents ejecting heat in the indoor environment. Following our previous work (Yan et al., 2016), using a simplified manikin model is unable to get an accurate CFD predictions. The prediction errors are found particularly jeopardize the manikin. In the present study, computational model using very fine 3D-scanned thermal manikin models (see Figure 8.4) was adopted to accurately capture the thermal response from passengers in the CFD simulations. Unfortunately, the finely spaced surface and volume mesh at the vicinity of the manikins have also dramatically increased the cost of computational resources. For example, for the given computational domain of this study, running a CFD steady state simulation using 2-core CPU at 2.9 GHz with the fine mesh requires roughly 37 hours to obtain a converged solution. For generating a sufficient training samples for the Kriging method, a total of 25 simulations are
required which translates into an approximately of 925 hours. It obviously poses a significant burden to the computational resources even with the modern parallel computing technology. In attempting to lower the computational time and cost, a Multi-Fidelity Kriging algorithm has been developed by using part of high-fidelity CFD results (fine mesh) and other part of low-fidelity CFD results (coarse mesh) as the training database.

To demonstrate the performance of the Multi-fidelity Kriging, a one-dimensional wave function with low and high-fidelity attributes is adopted for the assessment which is given by the following equations:

\[
\begin{align*}
    f_{HF}(x) &= 0.5\sin(10\pi x)/x + (x-1)^4 + 3\cos(\pi x) + 2\log(x) \\
    f_{LF}(x) &= 0.5\sin(10\pi x)/x + (x-1)^4 
\end{align*}
\]

Figure 8.14 An example of 1-D Multi-fidelity Kriging prediction results. (a) The prediction using only a few of high-fidelity data is undersampled and there are huge differences between the prediction values and the true values. (b) The prediction using a large amount of high-fidelity data is well sampled. (c) The prediction using a few of high-fidelity data plus a large amount of low-fidelity data achieves good prediction accuracy, where the low-fidelity data are inaccurate but help to correct the prediction.
Training samples were then extracted from the low and high-fidelity function as an input for Kriging process. Figure 8.14(a) shows the true function plot of the $f_{HF}$ (black solid line) in comparison to the Kriging prediction (red dash curve) with only 5 training samples. From the figure, one can observe that the Kriging predictions are a smooth function which pose considerably significant errors in compared with the true function. This prediction error is attributed to the under-sampled training data for the Kriging. The lack of information lead to a mis-interpretation of the true function in the Kriging process.

Obviously, the prediction accuracy can be improved by providing sufficient training samples (see as Figure 8.14 (b) with 20 samples). Meanwhile, the additional high-fidelity training samples would increase the computational cost. With the multi-fidelity Kriging, training samples can be extracted from the low-fidelity equation (i.e. $f_{LF}$) reducing the quest of computational burden. The key of the algorithm is to establish a correlation between low and high-fidelity samples.

Figure 8.14 (c) shows the Multi-fidelity Kriging prediction using only 5 high-fidelity training samples but adding 20 training samples from the low-fidelity equation $f_{LF}$. From the figure, one can observe that the accuracy of the Multi-fidelity Kriging could be improved significantly by replacing the high-fidelity samples with low-fidelity samples while saving substantial computational cost. The above assessment clearly demonstrated the advantage of the Multi-fidelity Kriging in correlating the high and low-fidelity training samples and its capacity in producing high accuracy predictions with limit computational cost.

8.4.5 Multi-fidelity Training Samples for HST cabin Ventilation Problem

In this study, the cabin geometry model was discretized using unstructured computational mesh by ANSYS ICEM. To accurately capture the VOCs dispersion, high quality fine meshes were generated around diffusers and 3D-scanned manikins. Furthermore, for resolving the boundary layer, inflation layers with a grid expansion
ration was applied around manikin surface. A grid sensitivity study was conducted over four different grid resolutions ranging from coarse mesh (i.e. ~2.8 million grids), good-mesh set (i.e. ~3.6 million grids), fine-mesh set (i.e. ~6.0 million grids) and extremely-fine-mesh set (i.e. ~7.7 million grids). The grid sensitivity study results of air flow velocity along the central line of the cabin were illustrated in Figure 8.15. As depicted, the fine-mesh predictions have less than 5% difference comparing the one from extremely-fine-mesh set. The result clearly demonstrated that the fine mesh adopted for all high-fidelity CFD training mesh are grid independent. Meanwhile, it is clear that the predictions of the coarse mesh pose considerably significant errors (i.e. some of the points are over 300%) compared to the fine and extremely-fine mesh. To demonstrate the robustness of the Multi-fidelity Kriging, predictions from the coarse mesh are adopted as low-fidelity training samples.

![Figure 8.15 Mesh independent testing results](image)

In this study, we adopted the fine-mesh set (5.9 million grids) and coarse-mesh set (2.8 million grids) to generate the multi-fidelity training cases. The distribution of the multi-fidelity cases is shown in Figure 8.16, where 9 fine mesh cases plus 25 coarse mesh cases were generated. It is noted that at the 9 locations of the fine mesh cases, the corresponding coarse mesh cases were also performed (i.e. the overlap points in Figure 8.16). With the CFD simulation results from all the multi-fidelity cases (34 in total), we can predict the response surface of the three design objectives using the Multi-fidelity
Kriging algorithm. Figure 8.17 shows the comparisons of the multi-fidelity predictions for the three design objectives in comparison to the predictions of the high-fidelity Kriging. As depicted, the predictions of the Multi-fidelity Kriging compares extremely well with the predictions of the high fidelity Kriging. The maximum difference between two Kriging is less than 5% which clearly demonstrates the accuracy and robustness of the Multi-fidelity Kriging.

Figure 8.16 The arrangement of low and high-fidelity training samples for Multi-fidelity Kriging

Figure 8.17 Comparison of prediction differences between Multi-fidelity Kriging and fully high-fidelity Kriging
The Multi-fidelity Kriging model is then coupled with the NSPSO for design optimization searching the optimal solution within the design space. Figure 8.18 shows the comparison of the Pareto Front obtain by using the high-fidelity (see all the blue dots) and Multi-fidelity Kriging (see all the red dots). The comparison clearly indicates that the Multi-fidelity Kriging is capable to yield reliable and accurate prediction for the NSPSO optimization procedure. The resultant Pareto front shares the overall characteristics and captures the quantitative values as the high-fidelity Kriging method. The maximum prediction error is maintained at 8.16% which is considerably low for ventilation design problem.

Figure 8.18 Comparison of optimal solutions ('Pareto Front') in objective space between using original high-fidelity Kriging and Multi-fidelity Kriging

One the other hand, the Multi-fidelity Kriging offers generous saving of the total computational time (i.e. the total of CFD simulation and the optimization computational time) comparing to the high-fidelity Kriging approach. A summary of
the total computational time (including optimization time) of the two approaches are tabulated in Table 8.2. With Multi-fidelity Kriging, 16 training samples extracted from the coarse mesh were selected in replacing the fine mesh training sample. At the same time, 9 training samples from the coarse mesh overlapped with the fine mesh samples were also included to establish the correlation between fine and coarse mesh CFD prediction. The table shows that using Multi-fidelity Kriging could reduce up to 35.61% of the total computational time which is over 1/3 of the total time required for the optimization procedures.

Table 8.2 Comparisons of CPU time consumptions (100 particles in the optimization process)

<table>
<thead>
<tr>
<th>Approach</th>
<th>Procedure</th>
<th>CPU Time (s)</th>
<th>Iterations</th>
<th>Total CPU Time (s)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Traditional</td>
<td>CFD simulation</td>
<td>1.332E+05</td>
<td>25</td>
<td>3.330E+06</td>
</tr>
<tr>
<td>NSPSO + Kriging</td>
<td>CFD (Fine mesh)</td>
<td>1.332E+05</td>
<td>25</td>
<td>3.330E+06</td>
</tr>
<tr>
<td></td>
<td>NSPSO</td>
<td>1.653E+02</td>
<td>1</td>
<td>1.653E+02</td>
</tr>
<tr>
<td>NSPSO + Multi-fidelity Kriging</td>
<td>CFD (Fine mesh)</td>
<td>1.332E+05</td>
<td>9</td>
<td>1.199E+06</td>
</tr>
<tr>
<td></td>
<td>CFD (Coarse mesh)</td>
<td>3.780E+04</td>
<td>25</td>
<td>9.450E+05</td>
</tr>
<tr>
<td></td>
<td>NSPSO</td>
<td>2.371E+02</td>
<td>1</td>
<td>2.371E+02</td>
</tr>
<tr>
<td>Total Saved CPU Time</td>
<td></td>
<td>1.186E+06(s)</td>
<td>(35.61%)</td>
<td></td>
</tr>
</tbody>
</table>

8.5 Conclusions

A multi-objective design approach has been proposed in this paper by incorporating the nondominated sorting based particle swarm optimization (NSPSO) algorithm with the Kriging method to find the optimal air supply conditions of the ventilation system in a
fully occupied high-speed train (HST) cabin for striking a balance among thermal comfort, contaminant control and energy consumption. The usage of NSPSO is able to provide multiple optimal solutions in one simulation run and gives a clear visualization of solutions in both design space and objective space. Users can easily select alternative solutions according to their experience and preference, rather than being struggled of choosing appropriate weightings at the beginning of traditional design process.

In order to validate the proposed approach, a high-resolution computational model of Chinese CRH2 train cabin was created with ANSYS Fluent package, where the fine 3D-scanned thermal manikin models were used to improve the computational accuracy. Multi-fidelity Kriging surrogate was developed in this study to address the time-consuming problem in CFD simulations, especially when we introduced the high resolution computational thermal manikin model. The simulation results demonstrate that the proposed new design procedure is capable to save up to 35.61% of the total computational time compared to the traditional single objective approach, while maintaining an acceptable predictive error.
Chapter 9

Conclusions

The holistic optimization of air quality in HST cabin is absent in existing literatures due to the extreme complexity of the impact factors, which are dependent with unknown relationships. The HST cabin environment is influenced by many factors, ranged from cabin interior to exterior. Considering the interior, diffuser design is key to the ventilation performance, which can induce the overall ventilation flow pattern in train cabin. Additionally, passengers' activities from interior cabin can also influence the air quality, such as coughing droplets from passenger which are the main source of contaminate transportation. On the other hand, train operation conditions and ambient environment are the main exterior impact factors for air quality. High pressure waves are induced in two situations: when crossing another HST and when passing through a tunnel. Though these two situations always happen during HST operation, the study on the impact of the induced pressure waves on HST interior air quality is rare. Besides, considering HSTs are significantly exposed to solar radiation, the change of interior air quality due to solar radiation cannot be neglected. In order to optimize the air quality in HST, it is necessary to consider various objectives such as thermal comfort, containment control and energy consumption. Therefore, an integrated optimization approach is expected. This thesis, however, further evaluated these overlooked factors associated with in-depth investigations and optimisation on both theoretical and numerical models. The main contributions from this thesis are:

- A comprehensive understanding of different type of diffuser effect on HST cabin ventilation performance, thermal comfort and containment dispersion processes.
A precisely cough-jet model to represent the transport and distribution of cough-generated airborne contaminants under the cabin environment.

A systematic CFD model of the interior air response to the induced pressure waves during the period of two HSTs passing by each other and the HST passing through a tunnel.

A quantizable approach to assess the solar radiation effect on thermal comfort in a HST cabin.

An efficient approach to manage the multi-objective optimization in HST cabin ventilation system.

The computational studies presented in this thesis lay a solid foundation for air quality optimization and health risk assessments in HST cabin environment, which can be also applied in other densely occupied spaces such as metro, bus and airline. Meanwhile, the outcomes of this study can be a supplement to the current industrial standards.

9.1 Summary of the Contributions

The summary of original contributions in each chapter of this thesis are:

9.1.1 The Effects of Diffuser Type on Thermal Flow and Contaminant Transport in HSR Cabins

Chapter 4 evaluates the importance of cabin diffuser effect in HST cabin environment, which is the key point to the cabin ventilation system. It employed a section model of CRH2 cabin containing four rows of seats with fully occupied passengers to investigate the effects of diffuser type on airflow field and contaminant transport characteristics. Simulations were conducted with consideration of cabin thermal load analysis and mathematic models were validated using experimental data from a mock-up cabin. Conclusions arose from this study are as follows.

- The geometry and location of diffusers have significant effects on ventilation performance and contaminate transportations in HST cabins. The overall and local airflow fields vary, while all PMV indexes are ideal. Similarly, contaminants distribution process varies, all four type diffusers have low concentration of
gaseous contaminants and can efficiently remove the particle contaminants in a certain time. If only considering breathing zone area, Diffuser type 3 has the best containment distribution performance, because the average and variance values of VOCs concentration and droplets volume fraction are the smallest among four types of diffusers.

- Gaseous contaminants and particle contaminants transport performances are different under the same ventilation condition. For instance, Diffuser type 1 has better VOCs distribution performance in passenger breathing zone compared with Diffuser type 4, but the particle distribution performance is worse.
- This study provides a more comprehensive way to assist assessing the diffuser efforts in ventilation schemes, in terms of not only thermal flow fields, but also contaminants removal, which could be a supplement to the current industrial standards.

9.1.2 The Effects of Cough-jet on Airflow and Contaminant Transport in a Cabin Section

Chapter 5 investigates the effect of cough-jet on local airflow and contaminant transport in a typical cabin environment by using CFD. Simulations are conducted by considering coughing as a transient generated process and one cough consists of different droplets size and number. Through comparing numerical results with experiment data, the prediction accuracy of the proposed numerical approach was validated. The conclusions yielded from this study are summarized as follows:

- Cough-jet has significant effects on air flow in front of cough-passenger in a short period time. In the time region of 0-0.5 second and the distance from mouth less than 0.45 meter, the cough-jet dominant the airflow. After this period, the effects of cough-jet gradually decrease, ended by the airflow come back to the steady state.

- Simulation results of droplet distribution by considering cough-jet model have significant difference from without considering cough-jet model. The most obvious difference was particles were carried away by ventilation flow immediately after released from mouth in without cough-jet model situation, while in applying
cough-jet model condition, particles were dominated by ventilation flow until the effects of cough-jet was decreased after a certain time and distance from mouth.

- Similar with a calm indoor environment, the distribution process of droplet was affected by the size in the cabin. For particles with diameter larger than 5E-5m, gravity caused by self-weight dominated the particles motion, while drug force caused by airflow dominated the motion of particles with size smaller than 5E-5m.

9.1.3 The Effects of Induced Pressure Fluctuations on Air Quality in HSR Cabins

Chapter 6 tests the response of cabin interior airflow, contaminant transport and thermal comforts to the transient pressure fluctuation when two HSTs passing by each other and when the HST passing through a tunnel.

For the situation that two HSTs passing each other, transient numerical simulations were conducted to analyse the effects of pressure fluctuations on the thermal environment and air quality in a CHR-2 HSR cabin. The HST cabin used in this study operates at a speed of 300 km/s. Two major pathways through which the exterior pressure waves intrude into to cabin, namely the door gaps and air-conditioning ducts, were considered. The computations revealed that the interior pressure field responded instantly to the exterior pressure change and can cause significant aural discomfort. However, the pressure fluctuation, despite its considerable magnitude, cannot build a pressure gradient that is large enough to change the airflow pattern in the cabin. As a result, neither the thermal flow field nor the contaminant dispersion pattern were significantly influenced by the instantaneous pressure fluctuations. It indicates that the pressure fluctuation induced by two HSTs crossing each other is not a major consideration when investigating the thermal comfort and air quality in HSR cabins.

For investigating the interior air response to the induced pressure wave when a HST passing through a 1km tunnel, a transient simulation case was conducted with exterior induced pressure wave intrude into cabin through door gaps and air-conditioning ducts.
Simulation results indicate that induced pressure wave has minor effects on the average temperature and velocity in the whole cabin. But obvious airflow changes could be found in the area near front and back door. As the result, in the area near doors, both thermal comfort and contaminant transport can be affected by the induced pressure wave. Also, the induced pressure wave had strong influence on the passengers’ hearing pressure which lasts for the same period as passing through tunnel. This study demonstrates that for the scenario that the HST passing through a 1km tunnel, the induce pressure wave may only influence the cabin ventilation performance, air quality and thermal comfort in the areas near the door. Thus, it is not a major consideration for the rest area of train cabin.

9.1.4 The Effects of Solar Radiation on Thermal Comfort in HSR Cabins

Chapter 7 focuses on mechanistic study of solar radiation effects on thermal comfort in a typical HST cabin. The effect of solar radiation was discussed in terms of airflow pattern, temperature distribution, and thermal comfort indices. Parametric studies with 7 different daytime hours were carried out. The effects of using the roller curtain were also studied. The following conclusions were made:

- The overall cabin air temperature, especially the temperature near passengers, was significantly increased after applying the solar radiation effects in the simulation. Such phenomenon resulted in an increase of average PMV value. The distribution of PD value in HST cabin come to be more uniform with the highest PD value changed from 25% to 19%.

- Passengers sitting next to the south-side windows had an obvious surface temperature change at different time of a day, especially in the leg and shoulder areas. The average cabin temperature increased in the morning and decreased in the afternoon. Similar change was seen on the average PMV values with a variate of 0.27. The average PPD and PD values also changed with time among 6.3-6.9% and 11-13% respectively. While the overall average thermal comfort indexes were
under thermal comfort standard, these indexes were out of comfort standard for the passengers exposed to the solar ray.

- A roller curtain could effectively reduce the cabin average temperature. Particularly, for passengers exposed to the solar ray, the surface temperature could be reduced by 10K with reasonable thermal comfort indexes achieved. To improve the ride comfort and reduce energy consumption in hot weather, using the curtain could be a good option.

### 9.1.5 Multi-objective Optimization of Air Quality in HSR Cabins

A multi-objective design approach for optimize the ventilation performance in a HST cabin has been proposed in Chapter 8. By incorporating the nondominated sorting based particle swarm optimization (NSPSO) algorithm with the Kriging method, the optimal air supply conditions in HST cabin was presented with a balance among thermal comfort, contaminant control and energy consumption. The usage of NSPSO can provide multiple optimal solutions in one simulation run and gives a clear visualization of solutions in both design space and objective space. Users can easily select alternative solutions according to their experience and preference, rather than being struggled of choosing appropriate weightings at the beginning of traditional design process.

Multi-fidelity Kriging surrogate was developed in this study to address the time-consuming problem in CFD simulations, especially when we introduced the high resolution computational thermal manikin model. The simulation results demonstrate that the proposed new design procedure is capable to save up to 35.61% of the total computational time compared to the traditional single objective approach, while maintaining an acceptable predictive error.

### 9.2 Summary of Simulation Models

The summary of all the simulation models applied in this project is presented in Table 9.1. It contains the information of computational geometry, computational resources used, convergence criterion in the simulations and computational time required for each
simulation case. Table 9.1 can be used as the reference to future researches on similar topic.

Table 9.1 Summary of simulation models

<table>
<thead>
<tr>
<th>Chapter</th>
<th>Model Geometry Size (mm)</th>
<th>Model Meshing Number</th>
<th>Solver Software</th>
<th>Computational Resource</th>
<th>Convergence Criterion</th>
<th>Computational Time (hours)</th>
</tr>
</thead>
<tbody>
<tr>
<td>4</td>
<td>4800<em>3100</em>2250</td>
<td>3.5 million</td>
<td>Ansys CFX</td>
<td>6-parallel-processor, 16 GB RAM</td>
<td>1.0E-06</td>
<td>6</td>
</tr>
<tr>
<td>5</td>
<td>1800<em>100</em>215</td>
<td>1.9 million</td>
<td>Ansys CFX</td>
<td>6-parallel-processor, 16 GB RAM</td>
<td>1.0E-05</td>
<td>69</td>
</tr>
<tr>
<td>6.1</td>
<td>19200<em>3100</em>225</td>
<td>10.2 million</td>
<td>Ansys CFX</td>
<td>40-parallel-processor, 64 GB RAM</td>
<td>1.0E-05</td>
<td>56</td>
</tr>
<tr>
<td>6.2</td>
<td>19200<em>3100</em>226</td>
<td>10.2 million</td>
<td>Ansys CFX</td>
<td>40-parallel-processor, 64 GB RAM</td>
<td>1.0E-05</td>
<td>120</td>
</tr>
<tr>
<td>7</td>
<td>19200<em>3100</em>227</td>
<td>8.6 million</td>
<td>Ansys Fluent</td>
<td>8-parallel-processor, 32 GB RAM</td>
<td>1.0E-04</td>
<td>6</td>
</tr>
<tr>
<td>8</td>
<td>4800<em>3100</em>2250</td>
<td>5.9 million &amp; 3.6 million</td>
<td>Ansys CFX</td>
<td>2-parallel-processor, 16 GB RAM</td>
<td>1.0E-06</td>
<td>37&amp;19</td>
</tr>
</tbody>
</table>
Bibliography


