NUMERICAL CHARACTERISATION OF CONTAMINANT TRANSPORT AND DISTRIBUTION IN AIRLINER CABINS

Yihuan Yan, B.M.E.
School of Engineering
College of Science, Engineering and Health
RMIT University

A thesis submitted for the degree of
Doctor of Philosophy
2017
Declaration

I certify that except where due acknowledge 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 approval research program; and, any editorial work, paid or unpaid, carried out by a third party is acknowledged.

Yihuan Yan
School of Engineering,
College of Science, Engineering and Health,
RMIT University
Acknowledgements

On the paper, this thesis summarised the research outcomes I achieved in the past four-year studies as a PhD student. Deep down, it is a precious outcome that cultivated by countless supports, encouragements and care from a lot of people. I could have never done any of this without the supports from my supervisors, families and friends.

I would like to firstly acknowledge my primary supervisor Prof. Jiyuan Tu for his brilliant supervision and mentoring. I would not have even stood a chance to start this project without his support and encouragement. During my whole PhD study period, he always inspired me to explore my potential and encouraged me to face challenges in a positive way. I really appreciate his sagacious guidance not only to my PhD study, but also to my life track.

I would particularly thank my second supervisor Dr. Xiangdong Li for his tremendous contributions to the work and countless supports to me. He was always willing to help, kept me on track and gave me his best advices whenever I needed. I have greatly benefited from his continuous support and guidance.

Also, I would like to thank all the colleagues Dr. Kiao Inthavong, Dr. Sherman C.P. Cheung, Dr. Sara Vahaji, etc. in the CFD group for their supports. It is my fortunate to have this amazing group of colleagues working at the similar field. They always shared ideas and new skills with me and gave valuable feedbacks on my work. I really appreciated all the helps and encouragements I received from them.

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

Last but not least, I would like to express my deepest gratitude to my family for their support, care and love. Their non-conditional love, trust and confidence on me mean everything to me. Thank you so much.
Publications during Candidature

Peer Reviewed Journal Publications:

   DOI: 10.1080/23744731.2015.1090254, **IF=0.871**

   DOI: 10.1177/1420326X16653500, **IF=1.181**

   DOI: 10.1016/j.enbuild.2016.06.030, **IF=4.067**

   DOI: 10.1007/s12273-016-0319-y, **IF=1.170**

   DOI: 10.1016/j.buildenv.2017.05.013, **IF=4.053**

   DOI: 10.1016/j.buildenv.2015.01.010, **IF=4.053**
   DOI: 10.1016/j.buildenv.2015.06.014, **IF=4.053**

   DOI: 10.1007/s12273-015-0217-8, **IF=1.170**

   DOI: 10.1177/1420326X16688049, **IF=1.181**

    DOI: 10.1016/j.enbuild.2017.04.070, **IF=4.067**


**Conference Publications:**


The following publications have been used as portion of the materials in this thesis:


DOI: 10.1016/j.buildenv.2015.06.014.
Contents

Abstract 1

1 Introduction 4
  1.1 Motivation .................................. 4
  1.2 Objectives .................................. 7
  1.3 Thesis Outline ............................... 8
  1.4 Contribution ................................. 9

2 Literature Review 11
  2.1 Particulate Contaminants Classifications .......... 11
  2.2 Influencing Factors of Particle Transport in Airliner Cabins .... 13
    2.2.1 The Thermal Effect of Human Body .......... 13
    2.2.2 Ventilation Scheme .......................... 14
    2.2.3 Heat Ventilation and Air-conditioning (HVAC) .... 16
    2.2.4 Release Location of Contaminants .......... 17
    2.2.5 Other Factors ............................... 17
  2.3 Passengers .................................. 18
    2.3.1 Impact of Passenger Bodies ................. 18
    2.3.2 Computational Thermal Manikins (CTMs) .... 19
  2.4 Mathematical Models ........................... 22
  2.5 Cabin Environments and Risk Assessments .......... 23

3 Methodology 26
  3.1 Project Breakdown ............................ 26
    3.1.1 Modelling of Cabin Models based on the Real Airliner Cabin Geometries ................. 26
    3.1.2 Investigations of Mathematical Models and Various Affecting Factors in Simplified Indoor Spaces .......... 27
    3.1.3 Optimisation of Computational Thermal Manikins ........ 28
<table>
<thead>
<tr>
<th>Section</th>
<th>Title</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>3.1.4</td>
<td>Test of Various Affecting Factors under Cabin Environment</td>
<td>28</td>
</tr>
<tr>
<td>3.1.5</td>
<td>Extending the Simulation Field into a Full Cabin Model</td>
<td>28</td>
</tr>
<tr>
<td>3.2</td>
<td>Gas-Phase Modeling</td>
<td>29</td>
</tr>
<tr>
<td>3.2.1</td>
<td>Governing Equations</td>
<td>29</td>
</tr>
<tr>
<td>3.2.2</td>
<td>Mass Conservation Equations</td>
<td>29</td>
</tr>
<tr>
<td>3.2.3</td>
<td>Momentum Equations</td>
<td>30</td>
</tr>
<tr>
<td>3.2.4</td>
<td>Energy equation</td>
<td>30</td>
</tr>
<tr>
<td>3.2.5</td>
<td>Turbulence Modeling</td>
<td>31</td>
</tr>
<tr>
<td>3.2.6</td>
<td>The RNG $k - \varepsilon$ Turbulence Model</td>
<td>32</td>
</tr>
<tr>
<td>3.3</td>
<td>Multi-Phase Modelling</td>
<td>34</td>
</tr>
<tr>
<td>3.3.1</td>
<td>The Eulerian-Eulerian Model</td>
<td>34</td>
</tr>
<tr>
<td>3.3.2</td>
<td>The Eulerian-Lagrangian Model</td>
<td>36</td>
</tr>
<tr>
<td>3.4</td>
<td>Simplification Approach of CTMs</td>
<td>37</td>
</tr>
<tr>
<td>4</td>
<td>Passengers’ Thermal Effects on Airborne Particle Transport and Distribution</td>
<td>40</td>
</tr>
<tr>
<td>4.1</td>
<td>Introduction</td>
<td>41</td>
</tr>
<tr>
<td>4.2</td>
<td>Numerical Procedures</td>
<td>42</td>
</tr>
<tr>
<td>4.2.1</td>
<td>Computational Domain and Boundary Conditions</td>
<td>42</td>
</tr>
<tr>
<td>4.2.2</td>
<td>Mathematical Models</td>
<td>46</td>
</tr>
<tr>
<td>4.3</td>
<td>Results and Discussion</td>
<td>47</td>
</tr>
<tr>
<td>4.3.1</td>
<td>Fluid Flow</td>
<td>47</td>
</tr>
<tr>
<td>4.3.2</td>
<td>Droplets Transport and Distribution</td>
<td>53</td>
</tr>
<tr>
<td>4.3.3</td>
<td>Droplets Transport and Assessment of Health Risks</td>
<td>55</td>
</tr>
<tr>
<td>4.4</td>
<td>Conclusion</td>
<td>58</td>
</tr>
<tr>
<td>5</td>
<td>Investigation of Particulate Matter (PM) Transport in Enclosed Spaces</td>
<td>59</td>
</tr>
<tr>
<td>5.1</td>
<td>Introduction</td>
<td>60</td>
</tr>
<tr>
<td>5.2</td>
<td>Mathematic Models</td>
<td>62</td>
</tr>
<tr>
<td>5.2.1</td>
<td>The Eulerian-Eulerian Model</td>
<td>62</td>
</tr>
<tr>
<td>5.2.2</td>
<td>Other PM Transport Models for Comparison</td>
<td>63</td>
</tr>
<tr>
<td>5.3</td>
<td>Numerical Procedures</td>
<td>65</td>
</tr>
<tr>
<td>5.4</td>
<td>Results and Discussion</td>
<td>67</td>
</tr>
<tr>
<td>5.4.1</td>
<td>Model Validation and Comparison</td>
<td>67</td>
</tr>
<tr>
<td>5.4.2</td>
<td>PM Transport Prediction</td>
<td>74</td>
</tr>
<tr>
<td>5.4.3</td>
<td>Computational Cost</td>
<td>77</td>
</tr>
</tbody>
</table>
5.5 Further Discussion ........................................ 78
5.6 Conclusion ........................................ 83

6 Effects of Computational Thermal Manikin (CTM) Simplification on CFD Simulations 85
6.1 CTM Simplifications Effects on Thermal Flow Field around Human Bodies ........................................ 86
6.1.1 Introduction ........................................ 87
6.1.2 Methodology ........................................ 88
6.1.2.1 Computational Domain and Boundary Conditions .... 88
6.1.2.2 The Computational Thermal Manikins (CTMs) .... 90
6.1.2.3 Numerical Procedures ................................ 93
6.1.3 Results and Discussions ................................ 96
6.1.3.1 Model Validation ................................... 96
6.1.3.2 Effects of CTM Simplification ........................ 98
6.1.4 Conclusions ........................................ 103
6.2 Comparisons of Simplified CTMs on CFD Predictions under Different Ventilated Room ........................................ 105
6.2.1 Introduction ........................................ 106
6.2.2 Numerical Methods .................................. 107
6.2.2.1 Computational Domain ............................ 107
6.2.2.2 Geometry of CTMs ................................ 108
6.2.2.3 Numerical Producers .............................. 110
6.2.3 Results and Discussion ................................ 111
6.2.3.1 Quiescent Condition (Case 1) ..................... 111
6.2.3.2 Displacement and Mixed Ventilations (Case 2 and 3) 115
6.2.4 Conclusions ........................................ 122

7 Development and Evaluations of Simplified Manikins in Cabin Environments 123
7.1 A Novel and Quantifiable Manikin Simplification Approach for Occupied Cabin Environments ........................................ 124
7.1.1 Introduction ........................................ 125
7.1.2 Methods ........................................ 127
7.1.2.1 The Mesh Decimating Algorithm .................. 127
7.1.2.2 The CFD Computations ............................ 131
7.1.3 Results and Discussion .............................. 133
List of Figures

2.1 Relative size chart of common air contaminants (Howie, 1990) ........................................... 12
2.2 Particle deposition in nasal cavity ................................................................. 13
2.3 Velocity vector and magnitude contours for manikin at inlet velocity of 0.2 m/s (Salmanzadeh et al., 2012) ................................................... 14
2.4 Room airflow distribution under different ventilation schemes (Rim and Novoselac, 2009) ................................................................. 15
2.5 Room airflow distribution under different ventilation schemes (Rim and Novoselac, 2009) ................................................................. 16
2.6 Schematic Diagram of the Boeing 737-300 Aircraft on Flight 2 from Hong Kong to Beijing (Olsen et al., 2003) ................................................... 17
2.7 Over-simplified CTMs by (a) Craven and Settles (2006), (b) Villi and De Carli (2014) and (c) Rai and Chen (2012) ............................................... 19
2.8 Human-like CAD models by (a) Kilic and Sevilgen (2008), (b) Ruzic and Bikic (2014) and (c) Zhang et al., (2012) ............................................. 20
2.9 3D-scanned manikins by (a) Sorensen and Voigt (2003), (b) Gao and Niu (2004) and (c) Martinho et al., (2012) ............................................. 20
3.1 Schematic view of breaking down the project into separate components ....................... 27
3.2 The mesh decimating algorithm by Garland and Heckbert (1997) .............................. 37
4.1 Schematic views of the airliner cabin ................................................................. 44
4.2 Comparison of the cabin model of this study (pink) and that by Liu et al (blue) (Liu et al., 2012) ................................................................. 45
4.3 Velocity vector field across Y-plane for isothermal and thermal case. ......................... 48
4.4 Comparison of airflow vectors between experimental data and computational results. ................................................................. 49
4.5 Velocity vector field across X-plane for isothermal and thermal cases. ......................... 50
4.6 Comparison of vertical velocities between experimental and computational results along selected lines. ................................................................. 51
4.7 Comparison of air velocities at selected positions. .................. 53
4.8 Droplets trajectories for isothermal and thermal cases. ............... 54
4.9 Droplets transport and distribution at various time steps. .......... 56
4.10 Average droplet residence time in the passengers’ breathing zones. . 57

5.1 The computational domain based on Chen et al. (2006)’s experimental setup. ................................................................. 65
5.2 Air velocity component profiles predicted with different turbulence models. ................................................................. 68
5.3 Effect of inlet concentration on the concentration profile alone Line 3 (10nm, t = 1800s). ................................................................. 69
5.4 PM velocity fields predicted by the E-E model and the E-L model (10µm, 800mg/m^3, 1800s) ................................................................. 70
5.5 PM concentration predictions vs. experimental data (10µm, 800mg/m^3, 1800 s). ................................................................. 71
5.6 PM concentration contours at t = 1800 s (10µm, 800mg/m^3, 1800s) . 73
5.7 Particle accumulation near the chamber bottom (by the E-L model, 10µm, 487µg/m^3, 120s). ................................................................. 74
5.8 Predictions of PM concentration evolution (by the E-E and the E-L models, 487µg/m^3, 10µm). ................................................................. 75
5.9 PM concentration evolution predicted by the E-E model (487µg/m^3, 0.2 µm). ................................................................. 76
5.10 Model setup for computations with thermal condition. ............... 78
5.11 The predictions of thermal plume ........................................... 79
5.12 Air velocity contour in a circular area 0.25 m above the manikin head 80
5.13 Typical PM distributions yielded by the (a) E-E model and (b) the E-L model ................................................................. 81
5.14 CN distributions predicted by the (a) E-E model and (b) E-L model . 83

6.1 The Computational domain based on the experimental setup of Licina et al. (2014)................................................................. 89
6.2 Simplified and smoothed model (CTM 2) from the original 3D scanned model. ................................................................. 90
6.3 Rebuilt manikin model (CTM 3) based on the skeleton structures. 91
6.4 Highly simplified model (CTM 4) based on the key dimensions of the 3D scanned model. ................................................................. 92
6.5 Comparison of overall velocity contour in front of the sitting manikin. 96
<table>
<thead>
<tr>
<th>Section</th>
<th>Description</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>6.6</td>
<td>Quantitative comparison of the air velocity at selected positions.</td>
<td>97</td>
</tr>
<tr>
<td>6.7</td>
<td>Comparison of velocity contour and velocity vector in the breathing zone.</td>
<td>98</td>
</tr>
<tr>
<td>6.8</td>
<td>Velocity profiles along the selected lines.</td>
<td>99</td>
</tr>
<tr>
<td>6.9</td>
<td>Effects of CTM simplification on prediction of velocity and temperature fields</td>
<td>100</td>
</tr>
<tr>
<td>6.10</td>
<td>Thermal plume regions (velocity &gt; 0.20m/s) at 25cm above manikin head.</td>
<td>101</td>
</tr>
<tr>
<td>6.11</td>
<td>Area of thermal plume region under various velocity ranges.</td>
<td>102</td>
</tr>
<tr>
<td>6.12</td>
<td>Computational domain with case 1: quiescent condition; case 2: displacement ventilation; and case 3: mixed ventilation.</td>
<td>108</td>
</tr>
<tr>
<td>6.13</td>
<td>The schematic views of the rebuilt manikin (RM) model (on the left), 3D scanned original manikin (OM) model (in the middle) and the simplified and smoothed manikin (SM) model (on the right).</td>
<td>109</td>
</tr>
<tr>
<td>6.14</td>
<td>Comparison of (a) the overall velocity contour in front of the sitting manikin; (b) the velocity contour and velocity vector at breathing zone between the experimental (left) and the simulation (right) results.</td>
<td>111</td>
</tr>
<tr>
<td>6.15</td>
<td>Positions of selected lines with side view (X-Z plane) and top view (X-Y plane).</td>
<td>112</td>
</tr>
<tr>
<td>6.16</td>
<td>Velocity profiles along selected lines.</td>
<td>114</td>
</tr>
<tr>
<td>6.17</td>
<td>Temperature contour for studied cases.</td>
<td>115</td>
</tr>
<tr>
<td>6.18</td>
<td>Velocity vectors predicted by different CTMs under the displacement (left) and mixed (right) ventilations.</td>
<td>116</td>
</tr>
<tr>
<td>6.19</td>
<td>Velocity profiles at selected lines under displacement ventilation.</td>
<td>117</td>
</tr>
<tr>
<td>6.20</td>
<td>Temperature profiles at selected lines under displacement ventilation.</td>
<td>118</td>
</tr>
<tr>
<td>6.21</td>
<td>Velocity profiles at selected lines under mixed ventilation.</td>
<td>120</td>
</tr>
<tr>
<td>6.22</td>
<td>Temperature profiles at selected lines under mixed ventilation.</td>
<td>121</td>
</tr>
<tr>
<td>7.1</td>
<td>The mesh decimating algorithm by Garland and Heckbert (1997).</td>
<td>128</td>
</tr>
<tr>
<td>7.2</td>
<td>Simplification of the laser-scanned CTM model through the mesh decimating algorithm with different numbers of iteration.</td>
<td>130</td>
</tr>
<tr>
<td>7.3</td>
<td>The computational domain based on Licina et al. (2014).</td>
<td>131</td>
</tr>
<tr>
<td>7.4</td>
<td>The effects of mesh coarsening on different CTMs (Line 1).</td>
<td>133</td>
</tr>
<tr>
<td>7.5</td>
<td>Numerical prediction of the overall airflow field.</td>
<td>134</td>
</tr>
<tr>
<td>7.6</td>
<td>Airflow field in the breathing zone.</td>
<td>135</td>
</tr>
<tr>
<td>7.7</td>
<td>Airflow in the heat-affected zone is sensitive to the CTM simplification.</td>
<td>136</td>
</tr>
</tbody>
</table>
7.8 Predictive error caused by CTM simplification. 137
7.9 CFD model of the airliner cabin section. 138
7.10 The predicted airflow fields in Plane 1. 139
7.11 Velocity profiles along lines in Plane 1. 139
7.12 Predicted squalene concentration in Plane 1. 140
7.13 Normalized squalene concentration in the passenger breathing zones. 141
7.14 Geometric information of CTMs. 147
7.15 Computational domain; (a) CTM-1; (b) Manikin model by (Licina et al., 2014). 150
7.16 Airflow field in the Y = 0 m plane (CTM-1). 153
7.17 Comparison of velocity and temperature profiles at selected points. 154
7.18 Comparison of velocity vectors at breathing zone. 155
7.19 Comparisons of velocity profiles along the horizontal and vertical lines in the breathing zone. 156
7.20 Global distribution of the contaminants released by CTM-1. 157
7.21 Normalised concentration at three selected lines. 158
7.22 Concentration contour at Plane 1 in the breathing zone. 160
7.23 Concentration contour at Plane 2 above the manikin head. 161
7.24 Future applications of tested CTMs (CTM-3 & CTM-4) in 4-rows train cabin. 162
7.25 Velocity profiles at selected planes (15 cm in front of the sitting manikins). 163
7.26 Normalised contaminant concentration at the plane across Passenger D. 164
7.27 Time dependent contaminants transport released by Passenger D at row-2. 166
7.28 Normalised contaminant concentrations at passengers’ breathing zones. 167

8.1 Computational model of cabin section and passengers. 175
8.2 Original 3D scanned manikin (left); simplified manikin using mesh-decimating approach (right). 176
8.3 Mesh independence of velocity field. 177
8.4 Sensitivity test of particle number along the longitudinal direction. 178
8.5 Velocity vector comparisons between numerical predictions and experimental measurements (Li et al., 2015a) at row-4. 182
8.6 Comparison of velocity profiles between numerical predictions and experimental measurements (Li et al., 2015a) at selected lines. 183
8.7 Velocity distribution at four selected planes in front of the passengers;
   a. Plane 1 (1st row), b. Plane 2 (3rd row), c. Plane 3 (5th row) and
d. Plane 4 (7th row) .......................................................... 184
8.8 Case studies with different index patients .................................. 185
8.9 Particle transport across the whole cabin domain ...................... 186
8.10 Particle travel distance along (a) longitudinal; (b) horizontal; (c) ver-
tical directions ................................................................. 187
8.11 Normalised concentration of exhaled particles (top view) ............. 188
8.12 Infection risks in passengers’ breathing zones ............................ 189
8.13 Particle trajectories in highly risked passengers’ breathing zones (case
   1) .......................................................... 190
## List of Tables

5.1 The computational costs by different models. ........................................ 78

6.1 Body surface areas and segment weighting factors of CTM models. ... 93

6.2 Percentage differences of BSAs between the original model and the other models. ................................................................. 93

6.3 Head load and heat flux of CTM models. ......................................... 94

6.4 Percentage difference of heat load between the original model and the other models. ................................................................. 95

6.5 Grid information for different CTM models. ............................... 95

6.6 Grid information for different CTMs. ........................................... 110

7.1 Quantification of the CTM simplification. ..................................... 130

7.2 Body surface areas and segment weighting factors of CTM models. ... 149

7.3 Comparison of boundary conditions on the manikin bodies. .......... 151
Nomenclature

\( \alpha \) Volume fraction

\( \dot{m} \) Mass flow rate

\( \Gamma \) Diffusion coefficient of the scalar

\( \lambda \) Thermal conductivity

\( \mu \) Dynamic viscosity

\( \mu_t \) Turbulent viscosity

\( \nu \) Kinematic viscosity

\( \nu_t \) Turbulent kinematic viscosity

\( \overline{U_i} \) Mean x-velocity

\( \phi \) Scalar variable for governing equation

\( \rho \) Density

\( \tau_p \) Effective particle diffusivity

\( \varepsilon \) Dissipation rate of turbulent kinetic energy

\( C_D \) Drag coefficient

\( C_\mu \) Turbulent viscosity relationship

\( C_{VM} \) Virtual mass coefficient

\( CTM \) Computational Thermal Manikin

\( D \) Diffusion coefficient

\( d_p \) Particle diameter
\( F_D \) Drag force

\( F_g \) Gravity force

\( F_{TD} \) Turbulent dispersion force

\( F_{VM} \) Virtual mass force

\( H \) Enthalpy

\( k \) Turbulent kinetic energy

\( P \) Pressure

\( PM \) Particular Matter

\( Pr \) Prandtl number

\( Q \) Air flow rate

\( Re \) Reynolds number

\( Re_p \) Particle Reynolds number

\( S \) Source term

\( T \) Temperature

\( u_{i}' \) Fluctuating x-velocity

\( u_{i,j,k} \) Velocity, x, y, z component
Abstract

The in-flight air quality of commercial airliners has drawn increasing attentions with the rapid annual-growth of air travellers. Contaminants inside airliner cabins could be released from multiple sources (coughing, sneezing, ozone production, etc.) and would suspend inside the cabin as particulate matters (PM). The focus of this project is to comprehensively and effectively assess the transport characteristics of particulate contaminants in airliner cabins and the corresponding infection risks of passengers.

Computational Fluid Dynamics (CFD) has been proven as a cost-efficient approach to analyse and optimise air quality in airliner cabin environments. However, the holistic simulation of airliner cabin is still absent in existing literature due to the extreme complexity of the cabin environment induced by the multi-scale, multi-coupling and non-linear transport characteristics. Theoretically, existing studies mostly relied on the Lagrangian model to depict particle transport in occupied airliner cabins due to its mechanistic coupling of air-particle interactions. Studies were mostly limited in reduced cabin section due to the factor that the Lagrangian tracking model is very time consuming and costly induced by the individual-tracking strategy. Numerically, due to the extreme complexity of the cabin environment, computational thermal manikin (CTM) models were arbitrarily simplified, causing the deficiency of passenger body features, and the passengers thermal effect was also eliminated. This could provide misleading information, particularly in passengers breathing zones, when evaluating the infection risks associated with particulate exposure. This research, however, further evaluated these overlooked factors associated with in-depth investigations and optimisation of theoretical and numerical models. By integrating mechanistic multi-phase flow models, novel manikin simplification approach and 3D dynamic characterisation of contaminant transport, a systematic and cost-efficient platform
was thereby developed for comprehensive assessment of air quality and particulate contaminant transport in airliner cabins.

The main body of this thesis was composed of nine chapters. In the first two chapters, research background and comprehensive literature review were summarised in conjunction with the highlighted research gaps found in the existing literature, followed by the research methodology in Chapter 3. The main research contributions were demonstrated from chapters 4 to 8. Chapter 4 evaluated the importance of passengers thermal effect in densely occupied cabin environment, which were mostly overlooked in existing studies due to the complexity of cabin environment. In Chapter 5, three mathematical models (Lagrangian, drift-flux and newly proposed Eulerian-Eulerian (E-E) models) were tested and compared in terms of the reliability and efficiency. A particle source in cell (PSI-C) method based program was developed using Matlab to convert particle trajectories into concentrations. The computational thermal manikins (CTMs) were optimised and simplified using various approaches in Chapter 6. The degree/level of applied simplification approaches were found uncontrollable. As a solution, a novel and quantifiable simplification approach based on the mesh decimating algorithm was developed and tested under cabin environment in Chapter 7. Chapter 8 demonstrated a systematic assessment of contaminants transport and infection risks in a large scale cabin environment. All the major components tested in the previous chapters were integrated to achieve comprehensiveness. Unsteady flow behaviour at the aisle region of the cabin was noticed in this study and verified by the collaborators experimental measurement. The PSI-C based program was further optimised using Mathematica to provide smooth concentration distributions. A quantifiable approach to assess infection risks was also proposed in this chapter. All the aforementioned contributions are concluded and highlighted in Chapter 9, followed by a list of all published publications during the PhD candidature period.

This research contributed to the following outcomes: (a) A novel and quantifiable manikin simplification algorithm was developed to reduce the computational costs without sacrificing accuracy. (b) Comprehensive descriptions of inter-phase mechanisms were achieved in a cost-efficient way using E-E model to realise fast predictions of the PM concentration in
airliner cabins. (c) An unique technique to convert particle trajectories to concentration was developed and optimised based on the PSI-C method. (d) A quantifiable approach to assess the infection risks in the airliner cabins was proposed. (e) A systematic platform was developed to holistically assess the infection risks in airliner cabin environment. The outcomes of this research laid an important and solid foundation for air quality optimisation and health risks assessment in other densely occupied spaces (high-speed rail, metro, etc.).
Chapter 1

Introduction

1.1 Motivation

Commercial airliners, as a vital public transport mode, have been carrying more than 3.5 billion people travelling over the world in 2015 and the number is forecasted to be more than doubled (7.4 billion) in 20 years (Tyler, 2015). When air travel is bringing convenience and efficiency to human being and boosting the global aviation industry, great concerns on the in-flight conditions are also raised in the meantime. Due to the high average occupancy of commercial flights, which was around 80% in 2015 and is still climbing (Gill, 2016), the cabin environments are usually very crowded, especially in the economy class, in which passengers are sitting very close to each other and have very limited space to move. Also, the airliner cabin environment is well-known as a low-humidity environment with relative humidity under 20% (Cui et al., 2014). Passengers travelling under such highly-occupied and low-humidity environment over a long period of time would inevitably feel uncomfortable or even sick, whereas they will not be able to leave at will during the flight. Therefore, increasing attentions have been drawn on the in-flight conditions, especially on the air quality, thermal comfort and disease transmissions.

During the flight, since passengers are all seated still and directly exposed to the immediate surroundings over a long time, the air quality and disease transmission are closely related to the local contaminants suspending in the air. In most in-service aircrafts, the mixing ventilation scheme is employed to minimise the contaminants level in the cabin, which supplies the air from diffusers located near the sides of cabin ceiling or baggage compartments and exhausts waste air through outlets near the floor level. Such ventilation design was expected to be able to suppress contaminants in a level lower than the passengers’ breathing zones. However, airflow circulations still
exist in some local regions, which would inevitably increase the exposure risks of passengers. Existing studies have found a wide variety of contaminants, such as airborne droplets, Oxidation products and Volatile Organic Compounds (VOCs) (Mangili and Gendreau, 2005; Coleman et al., 2008), which can be released or generated from multiple sources in the cabin environment.

Among various types of the contaminants, the infectious saliva/phlegm droplets released through coughing or sneezing by passengers has been emphasised in many epidemiology reports (Kenyon et al., 1996, Mangili and Gendreau, 2005), after the lessons taught from the global outbreaks of Tuberculosis (TB), Severe Acute Respiratory Syndrome (SARS) and Swine Influenza (H1N1) (Zhu et al., 2010). The airliner cabin environments are highly susceptible to be responsible for the spread of communicable diseases (Lindgren and Norback, 2005). Since the passengers are sitting densely in a limited and enclosed space, diseases containing infectious pathogens (such as influenza and tuberculosis) released by index patients through coughing or sneezing would cause direct person-to-person infections (Escombe et al., 2007). However, investigating or predicting the transport behaviours of these infectious droplets could be very challenging and time consuming, as the transmission of airborne diseases in airliner cabins revealed very strong non-linear characteristics during the protracted investigations of several SARS infection cases in 2003 (Olsen et al., 2003). The relative locations of the infected passengers to the index patient were found very randomly distributed in the cabins (Olsen et al., 2003). Therefore, as the perniciousness of the saliva/phlegm droplets has been widely raised, the knowledge of their transport characteristics in the cabin environment is crucially required to precisely predict the infection risks of every individual passenger.

The computational fluid dynamics (CFD) technique can be utilised as a promising tool to investigate and predict the contaminants transport and distribution in the airliner cabin environment, since CFD is not only able to provide full-scale simulations and visualisation of the transport processes in a cost-efficient way, but also capable of leading to an in-depth understanding of the complicated physical phenomena (Nielsen, 2015). Basically, two distinct approaches, namely the Lagrangian model and the Eulerian model have been employed in CFD simulations to simulate particulate transport in indoor spaces. Existing studies mostly relied on the Lagrangian model to depict particle transport in occupied airliner cabins due to its comprehensiveness. It has its unique advantage in whole-process description of particle movement from the injection point to the final destination. However, this approach cannot give direct prediction to the particle concentration as only particle dynamic equations are solved.
Despite the advantage and drawbacks of the Lagrangian model, simulating real airliner cabin environments is very challenging due to the extremely high computational cost induced by the extreme complexities of the physical/chemical phenomena inside the cabin and the large amount of occupants. As a compromise, several affecting factors, especially human body related factors, such as passengers’ thermal effect, were overlooked or even eliminated in existing studies to save the computational cost.

Passenger bodies, as the core factor of the cabin environment, were mostly treated only as passive objects subjected to the environment when investigating the contaminant transport and exposure in existing studies (Poussou et al., 2010; Isukapalli et al., 2013). However, in reality, human bodies are continuously in contact and interacting closely with their surroundings by serving as obstacles of ventilation airflow and exchanging heat and mass simultaneously (Melikov, 2015). Existing studies (Rim et al., 2009; Salmanzadeh et al., 2012) demonstrated that even for a single person (standing or sitting) in a large room, the thermal plume generated by body heat could carry particulate matters from the near-floor level into the breathing zone. Under densely-occupied cabin environment with a large number of passengers sitting very close to each other, the uprising thermal plume could be intensified and its impact on the overall flow field may be enlarged. In addition, due to the high cruise height in the upper troposphere or the lower stratosphere, the ozone level outside the aircraft is significantly elevated, up to hundreds of ppb (Newchurch et al., 2003), which would directly lead to a high ozone level in the cabin through the ventilation system. Recent studies reported that the ultra-fine particles and semi-volatile organic compounds yielded from the ozone reactions with human skin lipids (Gao et al., 2015) have been identified as long-ignored but serious health threats to airliner passengers (Zhang and Chen, 2007a). Therefore, it is essential to consider human bodies as not only the sink but the source of the contaminants.

In simulations, a wide variety of manikin models were employed in existing study to imitate the passengers in the cabin environment. Although manikin models with detailed body features are preferred to capture realistic airflow field and contaminant distribution adjacent to the passengers, it is apparently unpractical for multi-occupant cabin environment, in which a large number of passenger models (over 200) are involved. Compromise has to be made in order to balance the accuracy against the cost. It seems that more attentions were paid on the overall airflow and contaminant distributions only, while compromises were mostly made on the human bodies due to the complexity of body features (Park et al., 2015; Horikiri et al., 2015). Existing studies (Mazumdar et al., 2011) proved that the predicted contaminant concentration
field in an airliner cabin section was strongly affected by the CTM geometry. However, a criterion of how to simplify passenger models for a multi-occupant space is still absent and thereby some manikin simplifications for multi-occupants simulations reported in the literature were quite arbitrary (Rai and Chen, 2012). Therefore, in order to effectively predict contaminant transport in airliner cabins and to expedite the understanding of the interactions between human occupants and their surroundings through CFD simulations, it is crucial to develop the passenger models properly with certain criteria. This is particularly important when contaminants are released from the occupants.

As aforementioned, the cabin environment is very complex comparing to other indoor environments induced by the multi-scale, multi-coupling and non-linear characteristics of contaminant transport. It seems that the complexity and particularity of the cabin environment creates a solid barrier and makes the holistic investigations of airliner cabin insurmountable. Some studies chose to bypass the barrier through overlooking some affecting factors or over-simplifying the mathematical and geometrical models. This study, however, further evaluated these overlooked factors associated with in-depth investigations and optimisation of theoretical and numerical models. By integrating mechanistic multi-phase flow models, novel manikin simplification approaches and 3D dynamic characterisation of contaminant transport, a systematic and cost-efficient platform was thereby developed for comprehensive assessments of air quality and particulate contaminant transport in airliner cabins.

1.2 Objectives

CFD was employed as the tool to simulate the airflow and contaminants fields in conjunction with codes in Matlab and Mathematica to post-process the results and provide in-depth analysis. Simulations were conducted using the commercial CFD package Ansys CFX (ANSYS, 2015). The specific objectives of this thesis are:

1. To obtain fundamental knowledge of particulate contaminants transport in relation to mechanistic and physical environments

2. To develop a quantifiable and reliable approach for passenger model simplifications

3. To evaluate different mathematical models and to improve the efficiency and reliability for predicting contaminants transport under large scale cabin environment
4. To accurately predict the contaminant transport under multi-scale and multi-coupling conditions

5. To effectively assess air quality and infection risks in densely occupied cabins using CFD as the platform

1.3 Thesis Outline

This thesis is composed of nine chapters. The project motivation, background and objectives are introduced in this chapter. The focuses of the following chapters are outlined below:

Chapter 2 provides a comprehensive literature review in relation to the existing researches in the airliner cabin environment, including the case investigations during the SARS outbreak period. The affecting factors of particulate contaminant transport in airliner cabins are carefully reviewed and the applied passenger models in existing numerical studies are summarised. The existing mathematical approaches to predict the particle transport in indoor spaces are 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 whole project was firstly broke down into multiple components to individually investigate the major components (mathematical models, affecting factors in cabin environment, etc.) and obtain the fundamental knowledge of each component. Meanwhile, both mathematical and geometric models were optimised and simplified to achieve better computational efficiency. Eventually, all the tested components were integrated into large scale cabin environments to establish a platform for future investigations under similar densely occupied environments.

Chapter 4 evaluated the importance of passengers thermal effect in densely occupied cabin environment, which were mostly overlooked in existing studies due to the complexity of cabin environment. The Lagrangian model was employed to predict contaminants transport trajectories. The outcomes proved that the thermal plume effect generated by passengers body heat plays a pivotal role on the thermal airflow and the contaminants transport.

Chapter 5 tested and compared three mathematical models (Lagrangian, drift-flux and newly proposed Eulerian-Eulerian (E-E) models) in terms of the reliability and efficiency. A particle source in cell (PSI-C) method based program was developed using Matlab to convert particle trajectories tracked by Lagrangian model into
concentrations. The comparison results revealed that the E-E model has comparable accuracy to Lagrangian model and performed much better than the drift-flux model.

Chapter 6 tested and compared several commonly used CTMs by various simplification approaches in the micro-environment of human body and different ventilation schemes. Initial results indicated that surface smoothing approach was able to reduce considerable computational cost and remain reasonable accuracy, whereas other simplification approaches would cause significant errors in the simulations. The level of simplification was found as the main challenging when simplifying the CTMs.

Chapter 7 aimed to solve the difficulties found in the previous chapter by developing a novel and quantifiable approach to simplify the CTMs. This simplification approach was based on mesh-decimating algorithm and was tested under both indoor spaces and cabin environments, significant reduction of computational cost was achieved with good accuracy. Thus, this approach is highly recommended for future utilisation in densely occupied environment with large amount of CTMs involved.

Chapter 8 demonstrated a systematic assessment of contaminants transport and infection risks in large scale cabin environments. All the major components tested in the previous chapters were integrated to achieve comprehensiveness. Unsteady flow behaviour at the aisle region of the cabin was noticed in this study and verified by the collaborators experimental measurement. The PSI-C based program was further optimised in Mathematica to provide smooth concentration distributions using Lagrangian model. A quantifiable approach to assess infection risks was also proposed in this chapter.

Chapter 9 concludes and highlights all the major outcomes from chapter 4 to chapter 8.

1.4 Contribution

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

- A novel and quantifiable computational thermal manikin (CTM) simplification approach was developed to reduce the computational costs without over sacrificing accuracy.

- Comprehensive descriptions of inter-phase mechanisms were achieved in a cost-efficient way using E-E model to realise fast predictions of the PM concentration in airliner cabins.
• A unique technique to convert particle trajectories to concentration was developed and optimised based on the PSI-C method.

• A quantifiable approach to assess the infection risks in the airliner cabin environment was proposed.

• A systematic platform was developed to holistically assess the infection risks in airliner cabins.

The outcomes of this research laid an important foundation for assessment and optimisation of air quality not just in airliner cabins, but also in other densely occupied spaces (high-speed rail, metro, etc.).
Chapter 2

Literature Review

2.1 Particulate Contaminants Classifications

According to the existing respiratory studies (Howie, 1990; Escombe et al., 2007), a wide variety of contaminants have been identified in indoor spaces, including virus, smoke, pollen and etc., as demonstrated in Figure 2.1. These indoor contaminants can be mainly summarised into three categories: gaseous contaminants, particulate contaminants and biological contaminants (Austin et al., 2002). In terms of the size distribution, it can be noticed from the size chart that most indoor contaminants concentrate on the size range from 0.001 \( \mu m \) to 1000 \( \mu m \), which has nearly six orders of the magnitude difference between the smallest and the largest contaminants. Under such a diverse range of size distribution, it is almost impossible to filter out all the harmful contaminants through human’s nasal filtration system. Therefore, it is essential to investigate and identify the range of contaminants that can easily escape human’s nasal filtration system and go deeper into the lung.

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 (Kelly et al., 2004; Hsu and Chuang, 2012; Li et al., 2012). The deposition efficiency of the inhaled particles measured by Kelly et al. (2004) and Hsu and Chuang (2012) were summarised in Figure 2.2. By integrating different researchers’ outcomes into one figure, the overall deposition curve were thereby plotted to describe the relationship between particle deposition efficiency and size distribution. It can be noticed from the deposition curve that human’s nasal system filters more than 80% of the particles smaller than 1 \( nm \) and larger than 10 \( \mu m \). However, the deposition efficiency of the nasal system is significantly reduced when inhaled particles are between 10 \( nm \) and 5 \( \mu m \). The lowest deposition efficiency (only around 5%) occurs at the particle size range from 100 \( nm \) to 2.5 \( \mu m \), which was lately defined as PM2.5 (Chen et al.,
PM2.5 are more dangerous than other particles due to their extremely low deposition rate in human’s nasal cavity, while a number of harmful contaminants are within this range including viruses, bacteria, etc.

In cabin environment, the infectious virus released through coughing or sneezing of passengers is in the spotlight. Chao et al. (2009) concluded from their experimental measurements that the geometric mean diameter of contaminants from coughing was 13.5 \( \mu m \) with average release speed of 11.7 \( m/s \). Although the released droplets through coughing or sneezing seems much larger than PM2.5, most sputum droplets would quickly evaporate (mostly within half second depending on the relative humidity) and become droplet nuclei with average diameter of 3.5 microns (Redrow et al., 2011). As the perniciousness of the contaminants by coughing or sneezing was raised, attentions were mostly paid on the transport and distribution characteristics of the contaminants. Gupta et al. (2011) numerically investigated the distribution of contaminants released through different behaviour (i.e. coughing, breathing and talking). Their outcomes concluded that contaminants released by coughing of the in-
Figure 2.2: Particle deposition in nasal cavity

dex patient behaved similar as those from breathing, but the number is much higher. Li et al. (2014a) experimentally measured the contaminants transport in an aircraft cabin and distinguished the difference of the transport behaviour between the gaseous and particulate contaminants. They found that the gaseous contaminants were primarily affected by the airflow, while the particle distribution was affected by more factors. To gain deep understandings of the particulate contaminants transport, it is essential to identify the major influencing factors that would significantly affect the particle transport and distribution in airliner cabins.

2.2 Influencing Factors of Particle Transport in Airliner Cabins

2.2.1 The Thermal Effect of Human Body

Previous researches have proven that particulate contaminant transport and distribution in indoor environments could be affected by many factors. First of all, the buoyancy driven thermal plume generated by human bodies would have a major impact in relation to the airflow pattern as well as the particle distribution (Salmanzadeh et al., 2012). The effect of buoyancy driven thermal plume in the vicinity of a sitting, heated manikin was studied by Salmanzadeh et al. (2012). The following Figure 2.3 from their study provided a comparison of velocity vectors and contours with and with-
out consideration of buoyancy driven thermal plume. As can be noticed from Figure 2.3, the overall flow pattern significantly changed when the thermal plume effect was considered and thereby changed the particle transport. Salmanzadeh et al. (2012) believed that thermal plume effect in the vicinity of human body not only leads to a high concentration of suspended particle in breathing zone, but plays an important role in transporting particles entrained from the floor into human’s breathing zone in rooms with displacement ventilation system. Their study of thermal plume effect was based on a sitting manikin in an enclosed environment. However, in a typical cabin environment, with larger number of passengers involved in such enclosed environment, the effect of thermal plume could be dramatically enlarged, and thereby the flow pattern would also be completely different.

2.2.2 Ventilation Scheme

Also, contaminant transport and distribution have been known to be affected by the ventilation schemes (Rim and Novoselac, 2009). An experimental study was conducted by Rim and Novoselac (2009) to investigate airflow and pollutant distribution under various ventilation schemes and manikin arm movements. The results from
their experiment imply that the airflow distribution is very sensitive to the ventilation schemes, which also affects the exhaled particle distribution. As given in Figure 2.4, airflow was separated into two major cycles under forced convection air supply at ceiling, whereas in the case of low-momentum air supply at floor, the airflow quickly went up after leaving the diffuser due to the effect of thermal plume. In terms of commercial aircrafts, the cabin ventilation schemes could be varied with different manufacturers and models. This thesis mainly focuses on the medium-size airliners (e.g. Boeing 737 and Airbus A330), which are the most popular and widely employed commercial flights under current aviation industry (Chen et al., 2010).

Figure 2.4: Room airflow distribution under different ventilation schemes (Rim and Novoselac, 2009)
2.2.3 Heat Ventilation and Air-conditioning (HVAC)

Beyond the ventilation schemes, the heat ventilation and air-conditioning (HVAC) setup will also affect the air flow pattern as well as the particle transport (Khn et al., 2009). A generic cabin section of the Airbus A380 upper deck was built and the temperature field measurements and large scale particle image velocimetry were conducted in Khn et al. (2009)’s study. It has been found from their study that interaction between the supplied air jets, negative buoyancy forces acting on these air jets and interaction of thermal plumes with the supplied air jets, are all influencing the flow field inside the cabin. Khn et al. (2009) also argued that the impact of these effects differs considerably depending on the HVAC on the configuration and relative mass flow settings at the supply inlets.

![Room airflow distribution](image)

(a) Isothermal condition

(b) Cooling condition

Figure 2.5: Room airflow distribution under different ventilation schemes (Rim and Novoselac, 2009)
2.2.4 Release Location of Contaminants

The source or location of the contaminants would also be a pivotal factor affecting particle transport and distribution. Olsen et al. (2003) conducted a case investigation of the Severe Acute Respiratory Syndrome (SARS) transmission on aircraft during the period of global SARS outbreak in 2003. The following Figure 2.6 provides a schematic diagram of the investigated Boeing 737-300 in relation to the index patient and probable case of SARS. The distribution of infected passengers was more irregular and non-linear around the index patient, although the risk to passengers was greatest if they were seated a few rows in front of the index patient (Olsen et al., 2003). The greater concentration of passengers who became infected in front of the index patient than behind him may point to the role of coughing in transmission, causing a combination of aerosol and small-droplet spread. Olsen et al. (2003)’s investigation was very costly and time consuming due to the non-linear transport of particulate contaminants. Most importantly, the investigation was done a few days after people getting infected and the disease had already further spread. However, with the participation of Computational Fluid Dynamics (CFD) techniques, such disease transmission would be able to predict much quicker and more effectively, while infected people could be treated and isolated much earlier.

Figure 2.6: Schematic Diagram of the Boeing 737-300 Aircraft on Flight 2 from Hong Kong to Beijing (Olsen et al., 2003)

2.2.5 Other Factors

In addition, other factors such as passenger movements (Spitzer et al., 2010), passenger breathing under high occupant density (Hayashi et al., 2002) and flight crew movements (Poussou et al., 2010) would also have impacts on air flow and particle
transport in airliner cabin environments. Spitzer et al. (2010) concluded that the addition of motion from people has a significant impact on particle characteristics found within the human breathing zone, while Poussou et al. (2010) pointed out that the movement of human body should be considered when investigating contaminant transport.

Although the influencing factors of the particulate contaminants transport were carefully studied in indoor spaces, systematic investigations on these influencing factors in real-scale cabin environment were still inadequate. To save the computational cost, many studies (Rai et al., 2013; Yan et al., 2009b) used unrealistic passenger models and simplified mathematic models to conduct the simulation. Since the airliner cabin is a densely-occupied environment where passengers are sitting very close to each other, the errors caused by geometric and mathematical models are expected to be significantly enlarged and thereby need to be carefully evaluated.

2.3 Passengers

2.3.1 Impact of Passenger Bodies

Most existing studies treated human bodies as passive objects subjected to the environment when investigating the contaminant transport and exposure (Poussou et al., 2010; Isukapalli et al., 2013). However, in reality, human bodies are continuously in contact and interacting closely with their surroundings by serving as obstacles of ventilation airflow and exchanging heat and mass simultaneously, (Melikov, 2015). Thus, human bodies should also be regarded as the prime heat source of thermal buoyancy flows in most modern built environments. Rim et al. (2009) found that the thermal plume induced by human body metabolic heat plays an important role in transporting contaminants from near-the-floor level into the breathing zone. This uprising thermal buoyancy flow was even found to be responsible for the connection between skin disease and respiratory diseases (Lewis et al., 1969; Rim et al., 2009).

In addition, taught from the lessons of global outbreaks of transmissible diseases such as SARS and H1N1 flu, the non-linear transport and exhalation-inhalation characteristics of pathogen-carrying droplets in densely occupied indoor environment (e.g. airliner and train cabins) have become a major research concern (Rothman et al., 2006; Sze To et al., 2009). More recently, it was reported that the ultra-fine particles and semi-volatile organic compounds yielded from the ozone reactions with human skin lipids (Coleman et al., 2010; Gao et al., 2015) have been identified as long-ignored but serious health threats to office occupants, airliner passengers and
metro commuters (Wang and Waring, 2014; Zhang and Chen, 2007a). Oxidation products such as acetone, 6-methyl-5-hepten-2-one (6-MHO), geranyl acetone and hexanal can also be generated through the chemical reactions between ozone and human skin lipids, in which squalene, linoleic acid (LA) and oleic acid (OA) were the main precursors (Zeng et al., 2013; Thornberry and Abbatt, 2004). By interacting with the buoyancy driven thermal plume, gaseous contaminants concentration could be potentially lifted up and suspend in the occupants’ breathing zones. Therefore, it is essential to consider human bodies as not only the sink but the source of the contaminants.

2.3.2 Computational Thermal Manikins (CTMs)

To help assessing the human body involved indoor conditions and estimating the health risks associated with contaminant exposures, computational thermal manikins (CTMs) representing the human occupants have been widely employed in the CFD investigations under various indoor spaces (Sorensen and Voigt, 2003; Gao and Niu, 2004). A wide variety of CTMs have been reported in the literature, which can be classified into three categories:

1. Simple manikin models (Craven and Settles, 2006; Yan et al., 2009a; Mazumdar et al., 2011; Rai and Chen, 2012; Villi and De Carli, 2014), which use simple geometries (e.g. cylinders, spheres and rectangular blocks, etc.) and their combinations to represent human bodies, are the simplest approximation of the human occupants, as illustrated in Figure 2.7.

![Over-simplified CTMs](image)

Figure 2.7: Over-simplified CTMs by (a) Craven and Settles (2006), (b) Villi and De Carli (2014) and (c) Rai and Chen (2012)
2. Human-like CAD models (Kilic and Sevilgen, 2008; Ztek et al., 2010; Zhang et al., 2012; Seo et al., 2013; Ruzic and Bikic, 2014), which are built based on the human skeleton structures (Ruzic and Bikic, 2014) using CAD codes, have identifiable head, torso, arms and legs, as shown in Figure 2.8. However, detailed body features such as eyes, nose, fingers and toes are generally ignored.

![Figure 2.8: Human-like CAD models by (a) Kilic and Sevilgen (2008), (b) Ruzic and Bikic (2014) and (c) Zhang et al., (2012)](image)

3. 3D-scanned manikins (Sorensen and Voigt, 2003; Gao and Niu, 2004; Martinho et al., 2012), which are reconstructed from 3D scans of full-size dummies, have detailed body and facial features and are the most accurate representation of human occupants, as demonstrated in Figure 2.9.

![Figure 2.9: 3D-scanned manikins by (a) Sorensen and Voigt (2003), (b) Gao and Niu (2004) and (c) Martinho et al., (2012)](image)
Usually, CTMs with simple geometries require much lower computational cost by allowing coarser computational grid size. However, simple geometries may also lead to the loss of local airflow details near the CTM surfaces, even though the impact on the airflow in the bulk regions were reported to be less significant (Deevy and Gobeau, 2006). Detailed CTMs, on the contrary, are beneficial to improved predictive accuracy particularly in the vicinity of the model surfaces but demanding relatively high computational cost. Due to the limitation of current computational capacity, detailed CTMs are usually used to analyse the thermal comfort and micro-environment of a single person (Sorensen and Voigt, 2003) without considering comprehensive surrounding effects, whereas simplified CTMs are widely employed to investigate the ventilation, contaminant transport and exposure in multi-occupant indoor spaces such as airliner and train cabins (Poussou et al., 2010; Wang et al., 2014a). However, the question how to select a compromised CTM based on the specific requirements still remains unanswered, despite this issue has been widely recognized and some efforts (Yan et al., 2009a; Deevy and Gobeau, 2006) have been devoted to seek the answer. As a quantitative guideline to an appropriately simplified CTM is absent, the CTMs available in the literature differed a lot from each other and the development of these CTMs was quite arbitrary.

Using three CTMs with different resolution levels of body features, (Deevy and Gobeau, 2006) analysed the effects of CTM geometry on CFD simulations of airflow field in a ventilated room. They reported that for the bulk region, the simplified CTMs (Category 1 and 2) returned very similar air velocity fields in bulk region to that yielded from the CTM with detailed body features (Category 3). However, significant differences were found in the regions close to the CTM surface. This was consistent with the conclusion drawn by Seo et al. (2013), who also found that more precise results were obtained for the evaluation of thermal comfort when a detailed CTM was used. Therefore, it could be expected that simplified CTMs may be sufficient for predictions of the global flow field, while detailed CTMs would be preferred when the near-occupant regions or the occupants themselves are concerned. It should be noted that the scenarios of Deevy and Gobeau (2006) and Seo et al. (2013) were quite simple, i.e. unfurnished rooms containing a single occupant in the middle and excluded the interactions between multiple or moving occupants. In recent years, CFD has been widely utilised in relevant studies with the increasing concerns on the health risks associated with communicable diseases (e.g. SARS and flu, etc.) in public transport aircraft/vehicle cabins (Olsen et al., 2003; Furuya, 2007). In a densely occupied narrow indoor space such as an airliner cabin, the bulk region free from the occupant
effects could be very small, thus the human thermal plumes could overlap and the predicted results would be highly sensitive to the CTM geometry. Rai and Chen (2012) simulated ozone distribution in an airliner cabin section using two different CTMs and found that the predictive error of ozone concentration in the passenger breathing zone could be as large as 15%. Mazumdar et al. (2011) investigated the effects of passenger movement on contaminant transport in an airliner cabin. A rectangular block, a cylinder and a human-like block-set were used in their study to represent the moving passenger, respectively. Significant difference on the patterns of contaminant distribution were found in their study. Furthermore, due to the strong non-linear characteristics of contaminant/pathogen transport in aircraft/train cabins (Olsen et al., 2003), a full-cabin CFD model containing dozens or even hundreds of CTMs is often necessary in order to achieve an all-sided prediction. However, this would largely increase the computational cost. For such large-scale computations, it is not practical to use 3D-scanned manikins when considering the computational efficiency. Therefore, choosing appropriately simplified CTMs in densely-occupied cabin environment is crucial for optimising the computational efficiency and accuracy.

2.4 Mathematical Models

The computational fluid dynamics (CFD) technique has been proven to be an efficient approach in analysing transport of particulate matters in indoor environments as CFD is not only able to provide full-scale simulations and visualisation of the transport processes in a cost-efficient way, but also capable of leading to an in-depth understanding of the complicated physical phenomena. Basically, two distinct approaches, namely the Lagrangian approach and the Eulerian approach have been employed in CFD simulations particulate transport in indoor air. Both the Lagrangian approach and the Eulerian approach simulate the airflow using the same set of conservation equations, but use different methods to model particle movement through the air.

The Lagrangian approach, which tracks a number of representative particles separately through the air, is the most popular two-phase flow model for modelling PM transport and has its unique advantage in whole-process description of particle movement from the injection point to the final destination and vice versa. It also allows an integrated inter-phase coupling which could include various interacting mechanisms between the phases. For example, in most studies employing the Lagrangian model (Zhang and Chen, 2007b; Zhao et al., 2008), the drag force, the buoyancy force and additional forces were effectively incorporated so that a complete description of the
forces acting on the particles was achieved. However, this approach cannot give direct prediction to the particle concentration as only particle dynamic equations are solved. Additional post-process is required to calculate particle concentrations through the statistics of a large number of particle trajectories yielded from CFD computations. During the past years, the so-called sampling volume method (Zhang and Chen, 2007b; Salmanzadeh et al., 2012) and the kernel method (Chang et al., 2012b) have been developed to estimate particle concentration based on the Lagrangian CFD results. However, the stability and accuracy of these post-process procedures are still not satisfactory as reported by Chang et al. (2012a,b).

On the other hand, the Eulerian approach has gained relatively higher reputation on saving computational cost and simulating pollutant concentration, whilst it cannot predict particle motions or provide particle transport tracks. The Eulerian approach comes with different models. During the past years, some simplified Eulerian models have been utilised to model gas-particle flows in indoor environments, including the single fluid model by Zhang and Chen (2007b), the mixture model by Zhao et al. (2008) and the drift-flux model by Zhao et al. (2008) and Chen et al. (2006). Zhao et al. (2008) found that when compared with the mixture model, the drift-flux model has better accuracy since more mechanisms such as gravitational settling are included in the latter model. In fact, all of the aforementioned Eulerian models are simplified by assuming the gas-particle mixture as a pseudo fluid or treating the particle phase as a transportable scalar. This drawback actually makes the inter-phase actions between the phases could not be fully described. Therefore, a more comprehensive Eulerian model which is capable of describing the transport of each phase as well as the inter-phase actions is in demand.

2.5 Cabin Environments and Risk Assessments

In cabin environment, since passengers are mostly seated, disease transmission in airliner cabins are predominantly controlled by the transport of airborne particles and droplets (Beneke et al., 2011; Li et al., 2014b). Since the movement of airborne particles is largely controlled by the airflow field (Longest et al., 2004), properly designed ventilation systems have been mainly relied on in airliner cabins to minimize the exposure risks associated with airborne particles and droplets. For the same reason, the transport characteristics of airborne particles in airliners were mainly judged based on the airflow field (Mangili and Gendreau, 2005; Li et al., 2014a). The mixing ventilation scheme is currently widely employed in modern airliner cabins, which supplies air
from diffusers located near the cabin ceiling or baggage compartments and exhausts waste air through outlets near the floor level. Such ventilation design was expected to be able to suppress the contaminants in a level lower than the passengers’ breathing zone (Li et al., 2014a). However, airflow circulations still exist in some local regions, which inevitably increase the exposure risks.

When assessing the exposure dose related health risks in the cabin environment, existing studies mostly relied on the Eulerian-based concentration distribution of the droplets, to identify the high health hazard regions (Isukapalli et al., 2013). It is undoubtedly that the Eulerian-based approach can provide very fast 3D predictions of the contaminants concentration distribution, which is an important parameter when assessing the health risks because passengers sitting inside the high-concentration regions would usually have higher health risks. A most recent study conducted by You et al. (2017) employed the aforementioned Wells-Riley equation in conjunction with the Eulerian model to investigate the effect of the gaspers on the passengers exposure risks in a half-row cabin section. The Wells-Riley method was combined with the two-phase flow model when assessing the exposure risks in the cabin environment. In order to fit the Wells-Riley equation to the Eulerian model, they assumed that the exposure time was the same as the flight duration. However, the actual exposure time could be much less than the flight duration due to the cabin ventilation and is significantly different to every individual passenger, depending on the relative location to the index patient. The exposure time length in the Wells-Riley equation could be a critical parameter affecting the infection risks. Beyond that, the particulate phase is assumed to be a continuum in the Eulerian framework, which directly leads to the loss of some critical information, such as the time of particle residence in a given domain. This shortage makes the Eulerian model physically untrue when assessing the infection risks, since the infectious pathogens are always released in conjunction with the droplets or particles and they are sharing the similar transport characteristics.

Alternatively, the Lagrangian particle tracking model was also utilised in several numerical studies (Zhu et al., 2006; Gupta et al., 2011) due to its unique advantage in source-to-destination tracing of particle movement. Initial conditions of the released droplets/particles were also carefully in the existing studies. Gupta et al. (2011) numerically investigated the distribution of contaminants released through different behaviour (i.e. coughing, breathing and talking). They concluded that contaminants released by coughing of the index patient behaved similar as those from breathing, but the number is much higher. Chao et al. (2009) concluded that the geometric mean diameter of contaminants from coughing was $13.5 \mu m$ with average release
speed of 11.7 m/s. Although studies on initial conditions of the released particles are accumulating in the existing literature, investigations on the other key parameters (i.e. particle travelling distance and particle travelling time) were still inadequate in the cabin environment. Although studies on initial conditions of the released particles are accumulating in the existing literature (Gupta et al., 2011; Chao et al., 2009), investigations on the other key parameters (i.e. particle travelling distance and particle travelling time) were still inadequate in the cabin environment. Also, when providing detailed 3D characterised trajectories of the released particles, the Lagrangian model requires significantly high computational resources to track them. To save the computational cost, many studies (Ztek et al., 2010; Rai et al., 2013) used a reduced size of cabin section (3 rows or less) with unrealistic passenger models to imitate the cabin environment. Thus, the contaminants transport was significantly constrained by the computational domain and thereby the travel distance and time of contaminants could be misleading. Since airborne respiratory pathogens must reach the target infection site of the receptor to commence the infection, accurate predictions of the travelling distance and time of the infectious pathogens are crucial. Thus, it is necessary to apply an extended cabin section with adequate space and realistic passenger models with proper body features when assessing the transmission of airborne diseases. However, how to effectively and holistically assess the contaminants transport and related infection risks in cabin environment in a cost-efficient way is still a challenging task to accomplish.
Chapter 3
Methodology

3.1 Project Breakdown

As aforementioned, CFD simulation of real airliner cabin environments is very challenging due to the high computational costs induced by the extreme complexities of the physical/chemical phenomena inside the cabin and the large amount of occupants. It was extremely difficult to apply the most effective geometry models, optimised meshes, proper mathematical models and correct simulation setups at the beginning. The numerical outcomes could be less robust if simulations are directly conducted under real airliner cabin environment without evaluating the important factors of the cabin environment. Thus, the strategy of this research was to firstly break down the whole project into multiple components, followed by individual investigation of each major factor (affecting factors, CTMs, mathematical models, etc.) to obtain fundamental knowledge. Meanwhile, both mathematical and geometric models were further optimised or simplified to achieve better computational efficiency. Eventually, all the tested components were integrated into large scale cabin environments to establish a platform for future investigations under similar densely occupied environments.

3.1.1 Modelling of Cabin Models based on the Real Airliner Cabin Geometries

In order to simulate real and accurate cabin environments, the geometry of the cabin models was built based on the real dimensions provided by the manufactures. Some features inside the cabin such as the interval distance between each row, position of the ventilation system and the upper luggage rack areas were maintained during the modelling. A reduced cabin section including three rows of passengers and passenger
seats was firstly modelled at the beginning of the research. Gradually, an extended cabin section involving seven rows of passengers and passenger seats were built for further simulations.

### 3.1.2 Investigations of Mathematical Models and Various Affecting Factors in Simplified Indoor Spaces

Through literature review, a number of factors were identified as major influencing factors that would significantly affect the airflow pattern and particle transport in the cabin environment. However, most of these factors were only tested or measured through experimental studies conducted under a small-scale chamber. Therefore, it is crucial to test those factors under similar indoor environment first to validate the reliability of CFD simulations and then further test them in a real cabin environment. To obtain fundamental knowledge of particulate contaminants transport and distribution, it is important to test different mathematical models in terms of accuracy, efficiency and applicable range and thereby select and optimise the most proper one for investigating particle transport and distribution in multi-occupied cabin environment. It would be more effective and efficient if the mathematical models are firstly
tested under a simplified indoor environment. Thus, a single sitting manikin model was extracted from the cabin model and placed in a single chamber to run the simulations. A bonus effect of testing these factors in a simplified chamber with single manikin is that the geometry of the model and mesh size applied to discretise the domain can be relatively fine, so that the obtained results would be more accurate.

3.1.3 Optimisation of Computational Thermal Manikins

To conduct simulations into an extended and fully occupied cabin environment, a significant large amount of computational thermal manikins (CTMs) would be involved in the simulations (e.g. 42 CTMs are needed for a fully occupied sever-row cabin simulation). This would dramatically increase the computational cost and difficulty of simulations. Thus, it is necessary to reduce the computational cost and accelerate the simulations through simplifying the CTMs. However, simplifying manikin models would unavoidably cause numerical errors and over-simplifying models would significantly enlarge the errors. Therefore, it is essential to find a promising approach to simplify the manikin models and find the optimised level of simplifications.

3.1.4 Test of Various Affecting Factors under Cabin Environment

Since the cabin geometry is more complex than the simplified chamber and the occupant density in the cabin environment is much higher than other indoor environments, it is necessary to integrate those major affecting factors and test them under the cabin environment. Because the cabin geometry is approximately axially transitional periodicity and cross-sectional symmetry, a simplified cabin section containing 3 passengers and 3 seats could be applied as a typical but simplified cabin environment to test the combined effects of those affection factors with less computational cost.

3.1.5 Extending the Simulation Field into a Full Cabin Model

Once all the affecting factors and proper mathematical model have been tested and validated under a simplified cabin section, the simulation field would be extend into a larger cabin model containing seven rows of passengers and seats. Eventually, a full-scale cabin model would be applied to investigate contaminants transport and distribution.
3.2 Gas-Phase Modeling

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

3.2.1 Governing Equations

The governing equation of the continuous gas phase is obtained by identifying the fundamental principles based on: conservation of mass, Newton’s second law for the conservation of momentum and first law of thermodynamics for the conservation of energy. It can be expressed using three-dimensional Cartesian coordinate system (3.1) or the general form of transport equation (3.2).

\[
\frac{\partial \phi}{\partial t} + \frac{\partial [u \phi]}{\partial x} + \frac{\partial [v \phi]}{\partial y} + \frac{\partial [w \phi]}{\partial z} = \frac{\partial}{\partial x} \left[ \Gamma \frac{\partial \phi}{\partial x} \right] + \frac{\partial}{\partial y} \left[ \Gamma \frac{\partial \phi}{\partial y} \right] + \frac{\partial}{\partial z} \left[ \Gamma \frac{\partial \phi}{\partial z} \right] + S_\phi \tag{3.1}
\]

\[
\frac{\partial \phi}{\partial t} + (\vec{v} \cdot \vec{\nabla}) \phi = \vec{\nabla} \cdot (\Gamma \vec{\nabla} \phi) + S_\phi \tag{3.2}
\]

where \( \phi \) is a general fluid property, \( t \) represents time, \( \Gamma \) is a general diffusion coefficient and \( S_\phi \) is the source term. This transport equation can be translated as:

Rate of changing of \( \phi \) + Convection of \( \phi \) = Diffusion of \( \phi \) + Source term of \( \phi \) \hspace{1cm} (3.3)

3.2.2 Mass Conservation Equations

To achieve the conservation of mass, the scalar variable \( \phi \) is identified as the fluid density \( \rho \) in the governing equation, while the diffusion is discarded. The conservation of mass equation can be thereby rewritten below:

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

\[
\frac{\partial \rho}{\partial t} + \vec{\nabla} \cdot (\rho \vec{u}) = 0 \tag{3.5}
\]
3.2.3 Momentum Equations

To apply the Newton’s second law for the conservation of momentum, the scalar variable φ is identified as the fluid velocity $u$ and $\Gamma$ is replaced by the constant viscosity $\mu$, respectively. The force induced by the pressure $p$ is added into the source terms and the momentum equations can be rewritten from the transport equation using the Cartesian coordinate system as below:

X momentum:
$$\frac{\partial u}{\partial t} + u \frac{\partial u}{\partial x} + v \frac{\partial u}{\partial y} + w \frac{\partial u}{\partial z} = -\frac{1}{\rho} \frac{\partial p}{\partial x} + \frac{\mu}{\rho} \left[ \frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2} + \frac{\partial^2 u}{\partial z^2} \right]$$ (3.6)

Y momentum:
$$\frac{\partial v}{\partial t} + u \frac{\partial v}{\partial x} + v \frac{\partial v}{\partial y} + w \frac{\partial v}{\partial z} = -\frac{1}{\rho} \frac{\partial p}{\partial y} + \frac{\mu}{\rho} \left[ \frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2} + \frac{\partial^2 v}{\partial z^2} \right]$$ (3.7)

Z momentum:
$$\frac{\partial w}{\partial t} + u \frac{\partial w}{\partial x} + v \frac{\partial w}{\partial y} + w \frac{\partial w}{\partial z} = -\frac{1}{\rho} \frac{\partial p}{\partial z} + \frac{\mu}{\rho} \left[ \frac{\partial^2 w}{\partial x^2} + \frac{\partial^2 w}{\partial y^2} + \frac{\partial^2 w}{\partial z^2} \right]$$ (3.8)

The momentum equation can be also summarised using the compact equation 3.9:
$$\frac{\partial \vec{u}}{\partial t} + (\vec{u} \cdot \vec{\nabla}) \vec{u} = -\frac{1}{\rho} \vec{\nabla} p + \nu \vec{\nabla}^2 \vec{u}$$ (3.9)

3.2.4 Energy equation

For the energy equation induced by the first law of thermodynamics for the conservation of energy, $\phi$ is substituted by the temperature $T$, while $\Gamma$ is the thermal diffusivity. The governing equation can be rewritten as the following equations:

$$\frac{\partial T}{\partial t} + u \frac{\partial T}{\partial x} + v \frac{\partial T}{\partial y} + w \frac{\partial T}{\partial z} = \frac{k}{\rho C_p} \left( \frac{\partial^2 T}{\partial x^2} + \frac{\partial^2 T}{\partial y^2} + \frac{\partial^2 T}{\partial z^2} \right)$$ (3.10)

$$\frac{\partial T}{\partial t} + (\vec{u} \cdot \vec{\nabla}) T = \Gamma \vec{\nabla}^2 T + S_T$$ (3.11)

where, $k$ is the thermal conductivity, $C_p$ represents the thermal capacity and $S_T$ is the internal thermal source.
3.2.5 Turbulence Modeling

Turbulent flow is a flow regime that usually behaves in an irregular and chaotic manner. In turbulent flow, the fluid motion is continuously changing in both direction and magnitude. In CFD simulations, the turbulent flow can be modelled using the direct numerical solution (DNS) method, the Large-eddy simulation (LES) model or the Reynolds-Averaged Navier-Stokes (RANS) model. The DNS and LES models are preferable when detailed turbulent eddy information are required since the DNS method resolves all scales of turbulent eddies and the LES model resolves all the large eddies. However, these two models require significantly high computational cost even under a very simple computational domain. The RANS model gains a high reputation on modelling reasonably reliable turbulence flow in a cost-efficient way, which is suitable for very complex computational domain (e.g. the cabin environment). In the RANS model, the scalar variable \( \phi(t) \) is consisted of the mean variable \( \bar{\phi} \) and the fluctuating component \( \phi'(t) \):

\[
\phi(t) = \bar{\phi} + \phi'(t) \quad (3.12)
\]

The velocity components can be expressed as:

\[
u(t) = \bar{u} + u'(t) \quad (3.13)
\]

The intensity of the turbulence flow can be defined by the ratio of the fluctuating velocity and the mean velocity:

\[
I = \frac{u'}{\bar{u}} = \frac{\sqrt{\frac{1}{3} (u_x'^2 + u_y'^2 + u_z'^2)}}{\sqrt{(\bar{u}_x^2 + \bar{u}_y^2 + \bar{u}_z^2)}} \quad (3.14)
\]

in which the mean velocity is:

\[
\bar{u} = \frac{1}{t} \int_{t_0}^{t_0+t} u(x, y, z, t) dt \quad (3.15)
\]

By averaging each term and re-arranging the equation (3.15),

\[
\bar{u}'(t) = u(t) - \bar{u} = \frac{1}{t} \int_{t_0}^{t_0+t} u'(t) dt = \frac{1}{t} \int_{t_0}^{t_0+t} (u(t) - \bar{u}) dt = 0 \quad (3.16)
\]

After substituting 3.13 into the mass conservation equation 3.5 and averaging each term, the new mass conservation equation can be rewritten as:
\[ \nabla \cdot (\rho \bar{u}) = 0 \quad (3.17) \]

Similarly, by substituting Equation 3.13 into the continuity and momentum Equation 3.9 and averaging each term, it forms:

\[ \frac{\partial \bar{u}}{\partial t} + \bar{u} \cdot \nabla \bar{u} + \bar{u}' \cdot \nabla \bar{u}' = -\frac{1}{\rho} \nabla p + \nu \nabla^2 \bar{u} \quad (3.18) \]

The new term \( \bar{u}' \cdot \nabla \bar{u}' \) obtained from the averaged momentum Equation 3.17 can be rewritten in relation to the Reynolds stress:

\[ \bar{u}'_i \cdot \nabla \bar{u}'_i = -\frac{1}{\rho} \frac{\partial \tau_{ij}}{\partial x_j} = -\nu_t \frac{\partial^2 \bar{u}_i}{\partial x_j^2} \quad (3.19) \]

in which the Reynolds stress is defined by Equation 3.20 and can be expressed using the turbulence kinetic viscosity:

\[ R_{ij} = \tau_{turb} = -\rho \bar{u}_i' \bar{u}_j' = \rho \nu_t \frac{\partial \bar{u}_i}{\partial x_j} \quad (3.20) \]

By substituting Equation 3.19 back into Equation 3.18, the averaged momentum term can be rewritten as:

\[ \frac{\partial \bar{u}}{\partial t} + (\bar{u} \cdot \nabla) \bar{u} = -\frac{1}{\rho} \nabla p + (\nu + \nu_t) \nabla^2 \bar{u} \quad (3.21) \]

When the variable \( \phi \) is replaced by the temperature (T), similar to the mass of conservation and momentum equations, the energy equation can be written as:

\[ \frac{\partial T}{\partial t} + (\bar{u} \cdot \nabla) \bar{T} = (\Gamma + \Gamma_t) \nabla^2 \bar{T} + S_T \quad (3.22) \]

### 3.2.6 The RNG \( k - \varepsilon \) Turbulence Model

To simulate air turbulence in enclosed spaces, the Renormalisation group \( k - \varepsilon \) model has been widely employed in many existing studies (Isukapalli et al., 2013; Liu et al., 2012; Li et al., 2014b) due to its high reputations on modelling indoor airflow and pollutant transport. The RNG \( k - \varepsilon \) model was derived by Yakhot and Orszag (1986a) from the stand \( k - \varepsilon \) model. In the standard \( k - \varepsilon \) model (Launder and Spalding, 1974), in which \( k \) represents the turbulence kinetic energy and \( \varepsilon \) stands for the rate of dissipation of turbulent energy:

\[ k = \frac{1}{2} \bar{u}_i \bar{u}_i \quad (3.23) \]
\[ \varepsilon = \nu \frac{\partial u_i \partial u_i}{\partial x_j \partial x_j} \]  

(3.24)

where, the turbulence kinetic viscosity (or eddy viscosity) \( \nu_t \) is obtained from:

\[ \nu_t = C_\mu \frac{k^2}{\varepsilon} \]  

(3.25)

The general differential equation of the \( k - \varepsilon \) model is:

\[ \frac{\partial}{\partial t}(\rho \phi) + \nabla \cdot (\rho \vec{u} \phi - \Gamma \nabla \phi) = S_\phi \]  

(3.26)

Since the standard \( k - \varepsilon \) model is only applicable for fully turbulent flows (Chen, 1995), it requires very comprehensive wall functions to provide accurate prediction of air turbulence and thereby all the transport coefficients are fixed. Although these transport coefficients are obtained through experimental measurements, they are not universal (Chen, 1995). This would directly constrain the applicable range of the standard \( k - \varepsilon \) model and make the model less compatible under complicated environment. Yakhot and Orszag (1986a) derived the standard \( k - \varepsilon \) model using a statistical technique so called renormalisation group (RNG) methods. This method was used to develop a theory for the large scales, in which the effects of the small scales are represented by modified transport coefficients (Yakhot and Orszag, 1986b; Chen, 1995). While the RNG \( k - \varepsilon \) model remains the same form as the standard \( k - \varepsilon \) model, all the model coefficients were assumed as different values. A broader applicability could be thereby achieved for the RNG \( k - \varepsilon \) model, which is more reliable and accurate than the standard \( k - \varepsilon \) model under very complex environment with wider class of flows. The transport equation of the RNG \( k - \varepsilon \) model employed in the ANSYS CFX (ANSYS, 2015) is given below:

\[ \frac{\partial (\rho \varepsilon)}{\partial t} + \frac{\partial}{\partial x_j} (\rho U_j \varepsilon) = \frac{\partial}{\partial x_j} \left[ \left( \mu + \frac{\mu_j}{\sigma_{\varepsilon \text{RNG}}} \right) \frac{\partial \varepsilon}{\partial x_j} \right] + \frac{\varepsilon}{k} \left( C_{\varepsilon 1 \text{RNG}} P_k - C_{\varepsilon 2 \text{RNG}} \rho \varepsilon + C_{\varepsilon 3 \text{RNG}} \rho \varepsilon \right) \]  

(3.27)

\[ C_{\varepsilon 1 \text{RNG}} = 1.42 - f_\eta \]  

(3.28)

\[ f_\eta = \frac{\eta (1 - \frac{\eta}{4.67})}{1 + \beta_{\text{RNG}} \eta^2} \]  

(3.29)

\[ \eta = \sqrt{\frac{P_k}{\rho C_{\mu \text{RNG}} \varepsilon}} \]  

(3.30)
where, $\sigma_{\varepsilon\text{RNG}}$ is the RNG $k-\varepsilon$ turbulence model constant ($\sigma_{\varepsilon\text{RNG}} = 0.7179$), $C_{\varepsilon1\text{RNG}}$ is the RNG $k-\varepsilon$ model coefficient, which can be expressed by Equation 3.28 and $C_{\varepsilon2\text{RNG}}$ is the RNG $k-\varepsilon$ model constant ($C_{\varepsilon2\text{RNG}} = 1.92$).

### 3.3 Multi-Phase Modelling

#### 3.3.1 The Eulerian-Eulerian Model

In an Eulerian-Eulerian model, the particle phase is treated as an additional continuous phase inter-penetrating with the continuous air phase and two sets of conservations governing the balance of mass, momentum and energy of each phase are solved. As the inter-phase heat and mass transfers are not considered in this study, the conservation equations take the following form:

- **the continuity equation**

$$
\frac{\partial}{\partial t} (\alpha_i \rho_i) + \nabla \cdot (\alpha_i \rho_i \vec{U}_i) = 0 \tag{3.31}
$$

- **the momentum equation**

$$
\frac{\partial}{\partial t} (\alpha_i \rho_i \vec{U}_i) + \nabla \cdot \left( \alpha_i (\rho_i \vec{U}_i \vec{U}_i - \mu_i \left( \nabla \vec{U}_i + (\nabla \vec{U}_i)^T \right)) \right) = \alpha_i (S_{\text{Bouy}} - \nabla P_i) + \vec{F}_{ij} \tag{3.32}
$$

- **and the energy equation**

$$
\frac{\partial}{\partial t} (\alpha_a \rho_a H_a) + \nabla \cdot \left( \alpha_a (\rho_a \vec{U}_a H_a - \lambda_a \nabla T_a) \right) = 0 \tag{3.33}
$$

where, $i$ and $j$ are the phase denotations ($i, j = a$ for the air phase and $i, j = p$ for the particle phase). $\alpha$ is the volume fraction ($\alpha_a = 1 - \alpha_p$), $\rho$, $U$, $P$, $H$, $T$ and $\lambda$ represent the density, velocity, pressure, enthalpy, temperature and thermal conductivity, respectively. It should be noted that the energy equation 3.33 was solved only for the air phase while heat transfer within the particle phase was ignored in this study and thereby the heat transfer between two phases was not considered.

$S_{\text{Bouy}}$ is the momentum source due to buoyancy, which is defined in terms of a reference density $\rho_{\text{ref}}$ which takes value of air density at the inlet:

$$
S_{\text{Bouy}} = (\rho_i - \rho_{\text{ref}}) g \tag{3.34}
$$

When calculating the thermal buoyancy force induced by the thermal plume, the Buossinesq approximation is employed in the momentum equation to take into account thermal expansion of the air:
\[ \rho_a = \rho_{ref} (1 - \beta (T_a - T_{ref})) \]  

(3.35)

where, \( \beta \) is the coefficient of thermal expansion and \( T_{ref} \) is the reference temperature which takes value of air temperature at the inlet.

\( \vec{F}_{ij} \) in the momentum equation 3.32 represents the interfacial forces, which is formulated based on the assumption of spherical particles. The forces that may be significant include the drag force \( \vec{F}_D \), the turbulent dispersion force \( \vec{F}_{TD} \) and the virtual mass force \( \vec{F}_{VM} \), which are defined in the following equation 3.37,3.38 and 3.39, respectively.

\[
\vec{F}_{ap} = -\vec{F}_{pa} = \vec{F}_D + \vec{F}_{TD} + \vec{F}_{VM} \quad (3.36)
\]

\[
\vec{F}_D = \frac{3}{4} C_D \alpha_p \rho_a \left| U_p - U_a \right| \left( \vec{U}_p - \vec{U}_a \right) \quad (3.37)
\]

\[
\vec{F}_{TD} = -C_{TD} \rho_a k_a \nabla \alpha_a \quad (3.38)
\]

\[
\vec{F}_{VM} = C_{VM} \alpha_p \rho_a \left( \frac{d\vec{U}_p}{dt} - \frac{d\vec{U}_a}{dt} \right) \quad (3.39)
\]

where, \( C_D \) is the drag coefficient correlated to the particle Reynolds number, \( C_{TD} \) is the turbulent dispersion coefficient and \( C_{VM} \) is the virtual mass coefficient.

Although the inter-phase forces are formulated in a mechanistic way, the coefficients (e.g. \( C_D \), \( C_{TD} \) and \( C_{VM} \)) are generally determined empirically. For the spherical particles, the turbulent dispersion coefficient was modelled according to Lopez de Bertodano (1991) and a constant \( C_{VM} = 0.1 \) was employed for the virtual mass coefficient (ANSYS, 2015). The drag coefficient \( C_D \) is a function of the particle Reynolds number \( Re_p \) and was modelled in this study using the Florin (1978) correlation, which is valid for \( Re_p \leq 2000 \).

\[
C_D = \frac{24}{Re_p} \left( 1 + 0.149 Re_p^{0.687} \right) \quad (3.40)
\]
3.3.2 The Eulerian-Lagrangian Model

When an Eulerian-Lagrangian model is employed for a gas-particle flow, the air phase is still governed by the Eulerian equation (equations 3.31, 3.32, 3.33 with $\alpha_a = 1$, which means the volume fraction occupied by the particles is negligible.). The particles are tracked using the Lagrangian method separately through the airflow field. In the Eulerian-Lagrangian model, the particles are tracked using the equation of motion. For a spherical particle with diameter of $d_p$ immersed in continuous air, the drag force $F_D$, the buoyancy force $F_{Buoy}$ and the virtual mass force $F_{VM}$ are considered in order to keep the same inter-phase momentum transfer mechanisms as the Eulerian-Eulerian model.

$$m_p \frac{d\vec{U}_p}{dt} = \vec{F}_D + \vec{F}_{Buoy} + \vec{F}_{VM} \quad (3.41)$$

$$\vec{F}_D = \frac{C_D \pi d_p^2}{2} \rho_a \left| \vec{U}_p - \vec{U}_a \right| (\vec{U}_p - \vec{U}_a) \quad (3.42)$$

$$\vec{F}_{Buoy} = \frac{\pi d_p^3}{6} (\rho_p - \rho_a) g \quad (3.43)$$

$$\vec{F}_{VM} = \frac{C_{VM} \pi d_p^3}{2} \rho_a \left( \frac{d\vec{U}_p}{dt} - \frac{d\vec{U}_a}{dt} \right) \quad (3.44)$$

Being different to the Eulerian-Eulerian model, the effect of turbulent dispersion on particle transport is modelled in the Lagrangian model by adding an eddy fluctuating component onto the mean air velocity. It is the fluctuating component of the air velocity which causes the dispersion of particles in turbulent flow.

$$\vec{U}_a = \overline{U}_a + U' \quad (3.45)$$

In each eddy, the fluctuating eddy velocity can be varied by the lifetime $t_e$ and the length $L_e$ of the eddy. The impact of the fluctuating eddy velocity on the particles is only valid when the following two conditions are met. Firstly, the interaction time between the entering particle and the eddy is shorter than the eddy lifetime. Secondly, the relative displacement of the particle to the eddy is less than the eddy length. If not, the fluctuating eddy velocity in this eddy is not considered and the particle is assumed to be directly entering into the next eddy with new lifetime, length and thereby the new fluctuating velocity.
\[ U' = \Phi \left( \frac{2k^3}{3}\right)^{0.5} \]  

(3.46)

\[ L_e = \frac{C_{\mu}^{3/4} k^{3/2}}{\varepsilon} \]  

(3.47)

\[ t_e = \frac{L_e}{\left( \frac{2k^3}{3}\right)^{0.5}} \]  

(3.48)

where \( \Phi \) is a normal distributed random number which accounts the randomness of turbulence by a mean value. \( k \) and \( \varepsilon \) are the local turbulent kinetic energy and dissipation, respectively. \( C_{\mu} \) is the turbulent constant.

### 3.4 Simplification Approach of CTMs

Garland and Heckbert (1997) proposed a mesh decimating algorithm based on quadric error metrics, which could significantly simplify a complex geometry while preserving the primary features of the object. According to Garland and Heckbert (1997), a 3D geometry surface could be represented by a number of triangular faces. The basic idea of mesh decimation is to contract the pairs of triangle vertices, as illustrated in Figure 3.2. The operation of pair contraction \(((v_1, v_2) \rightarrow v)\) moves the vertices \( v_1 \) and \( v_2 \) to a new position \( v \), rebuilds the triangles by connecting all their incident edges to \( v \) and then deletes \( v_1 \) and \( v_2 \).

![Mesh Decimation Algorithm](image)

(a) Edge contraction

(b) Non-edge contraction

Figure 3.2: The mesh decimating algorithm by Garland and Heckbert (1997)
The key job of the mesh decimating algorithm is to determine an optimal position of \( v \). As shown in Figure 3.2, the new vertex \( v(x, y, z) \) and the original triangular planes \( p_i(ax + by + cz + d = 0) \) could be expressed in the form of matrices by:

\[
v = (x, y, z)^T \tag{3.49}
\]

and

\[
p_i = (a, b, c, d)^T \tag{3.50}
\]

Thus, the sum of squared distances of \( v \) to the planes \( (i = 1 - N) \) is:

\[
\Delta(v) = v^T \left( \sum_{i=1-N} K_{pi} \right) v \tag{3.51}
\]

where, \( K_{pi} \) is the matrix:

\[
K_p = p_i p_i^T = \begin{bmatrix}
a^2 & ab & ac & ad \\
ab & b^2 & bc & bd \\
ac & bc & c^2 & cd \\
ad & bd & cd & d^2
\end{bmatrix} \tag{3.52}
\]

This fundamental error quadric \( K_p \) can be used to find the squared distance of any point in space to the plane \( p_i \). For the planes \( (i = 1 - N) \) as shown in Figure 3.2, the sum of their fundamental error quadric makes a new single matrix \( Q \). Therefore, equation 3.51 is rewritten by

\[
\Delta(v) = v^T Q v \tag{3.53}
\]

In order to minimize the error, the position of \( v \) should satisfy:

\[
\begin{bmatrix}
q_{11} & q_{12} & q_{13} & q_{14} \\
q_{12} & q_{22} & q_{23} & q_{24} \\
q_{13} & q_{23} & q_{33} & q_{34} \\
0 & 0 & 0 & 1
\end{bmatrix}
v = \begin{bmatrix}
0 \\
0 \\
0 \\
1
\end{bmatrix}, \quad \begin{bmatrix}
q_{11} & q_{12} & q_{13} & q_{14} \\
q_{12} & q_{22} & q_{23} & q_{24} \\
q_{13} & q_{23} & q_{33} & q_{34} \\
0 & 0 & 0 & 1
\end{bmatrix}^{-1} \begin{bmatrix}
0 \\
0 \\
0 \\
1
\end{bmatrix} \tag{3.54}
\]

This mesh decimating algorithm was applied to simplify laser-scanned manikins in this study. The original manikin model with detailed body features was downloaded from the open database http://www.cfd-benchmark.com. The simplification was iteratively performed, with the following criteria of judging a valid vertex pair \((v1, v2)\) for contraction (Garland and Heckbert, 1997):
1. \((v_1, v_2)\) is an edge of a triangle (Figure 3.2a), or

2. \(\|v_1 - v_2\| < t\), where \(t\) is a threshold parameter for non-edge pair contraction (Figure 3.2b).

To begin with, the manikin model was divided into 250,000 initial triangular faces, which were sufficiently fine to fulfil an accurate capture of the dummy geometry. Then, a target percentage of reduction \((\Phi = 0.8)\) was set for each iteration in order to achieve a smooth simplification. Thus, the CTM simplification could be quantified using a dimensionless simplification index:

\[
SI = \frac{1}{N_0 \Phi^n}
\]  

where, \(N_0\) is the initial number of the triangular faces and \(n\) is the iteration number of CTM simplification. \(SI\) indicates the ratio of the mean area of a single triangular face to the total CTM surface area. Obviously, an elevated \(SI\) means larger triangular faces. Through controlling \(SI\) or the iteration number, the mesh decimating algorithm provides a quantitative and controllable approach to simplify the CTMs.
Chapter 4
Passengers’ Thermal Effects on Airborne Particle Transport and Distribution

The main findings of this chapter have been published in:


This paper numerically investigated the effects of the buoyancy-driven thermal plume on the airflow pattern and transport characteristics of airborne particles in airliner cabins. A cabin section containing 3 seats and 3 passengers were built and numerical simulations were conducted using thermal and isothermal conditions, respectively. Airborne particles were assumed to be released by the passengers through coughing. The predicted airflow field was validated using experimental data available in the literature. Comparison of the computational results revealed that the thermal plume changed significantly both the airflow field and the trajectories of particle transport. In addition, the spatial distribution characteristics of the particles and their residence time in the passengers’ breathing zones were highly sensitive to the location of released particles. Comparatively, the particles released by the passenger seated close to the window may have the highest health risk to other two passengers.
4.1 Introduction

Commercial airliners, as a vital part of the modern society owing to the continuous technological and economic advancements, have been carrying nearly two billion passengers travelling over the world every year (Poussou et al., 2010). However, as air travel brings convenience and efficiency to the humans, it is also hastening the global spread of infectious diseases. Airliner cabin environments have been highly susceptible to be responsible for spread of communicable diseases (Lindgren and Norback, 2005; Coleman et al., 2008). Recalling the global outbreaks of SARS in 2003, H1N1 flu in 2009 and a series of Mycobacterium Tuberculosis (TB) outbreaks during commercial flights (Abubakar, 2010; Yin et al., 2012), a great concern on the transmission of infectious diseases in the medium-size airliner cabins has been raised (Rothman et al., 2006; Bennett et al., 2013).

When passengers are all seated, disease transmission in airliner cabins are predominantly controlled by the transport of airborne particles and droplets (Beneke et al., 2011; Li et al., 2014a). Since the movement of airborne particles are largely controlled by the airflow field (Longest et al., 2004), properly designed ventilation systems have been mainly relied on in airliner cabins to minimise the exposure risks associated with airborne particles and droplets. For the same reason, the transport characteristics of airborne particles in airliners were mainly judged based on the airflow field (Mangili and Gendreau, 2005; Li et al., 2014a). Nowadays, the mixing ventilation scheme has been widely employed in modern airliner cabins, which supplies air from diffusers located near the cabin ceiling or baggage compartments and exhausts waste air through outlets near the floor level. Such ventilation design was expected to be able to suppress the contaminants in a level lower than the passengers’ breathing zone. However, airflow circulations still exist in some local regions, which inevitably increase the exposure risks. During the past decade, a number of investigations have been conducted with the aim to improve the airflow pattern in airliner cabins with mixing ventilation systems. Previous studies (Zhang et al., 2005; Khn et al., 2009; Poussou et al., 2010; Spitzer et al., 2010) have proven that the airflow field as well as the transport and distribution characteristics of particulate contaminants in an airliner cabin could be affected by many factors including the ventilation scheme, the blockage by seats and passengers, the movement of passengers and crews and the manikin thermal plume. Among them, the thermal buoyancy flow driven by the passenger body heat, known as the human thermal plume, is one of the most important factors (Nilsson et al., 2000). Existing studies (Rim and Novoselac, 2009;
Salmanzadeh et al., 2012) demonstrated that even for a single person (standing or sitting) in a large room, his/her thermal plume could carry particulate matters from the near-floor level into the breathing zone. As an airliner cabin is generally a densely occupied indoor environment with a large number of passengers sitting very close to each other (Lippmann et al., 2002), the uprising thermal plume could be intensified and its impact on the overall flow field may be enlarged.

On the other hand, as revealed by our previous studies of particle transport around human bodies (King Se et al., 2010; Ge et al., 2013; Li et al., 2013) and by Wang et al. (2014a)’s CFD simulation regarding contaminant transport in a high-speed train cabin, the trajectories of particle transport and its spatial distribution present very strong local features. Even for a given airflow field, a small change in the location of particle release would lead to a totally different particle inhalation and exposure risk (King Se et al., 2010; Ge et al., 2013; Li et al., 2013). Therefore, the airflow field only may not be an appropriate representation of the transport and distribution of airborne particles due to their different aerodynamic properties. Thus, it is crucial to investigate the characteristic of particulate contaminants transport individually, with the simultaneous consideration of the effects impacted by the airflow field. Unfortunately, to the best of our knowledge, quantitative studies on this issue are still very rare in the open literatures.

Therefore in this study, the transport characteristics of airborne particles produced by passenger cough in an airliner cabin section were numerically investigated using computational fluid dynamics (CFD). Emphasis was put on the effects of passenger thermal plume on the airflow fields and the characteristics of particle transport. The predicted airflow fields were visualised and compared against the experimental data available in the literature. The effects of passenger thermal plume on the transport and distribution characteristics of airborne particles were also analysed quantitatively by comparing the numerical results yielded from the thermal and isothermal conditions.

4.2 Numerical Procedures

4.2.1 Computational Domain and Boundary Conditions

By using particle image velocimetry (PIV) and hot-wire and hot-film anemometers, Liu et al. (2012) conducted an elaborate measurement of the airflow fields in the first-class cabin of a MD-82 commercial airliner. Their experimental measurements provided very detailed experimental data including the bulk airflow fields and the
ventilation jet profiles, which can be further applied for validating the computational fluid dynamics (CFD) models. By using electrically heated manikins, Liu et al. (2012) also investigated the effects of passengers on the overall airflow pattern in the airliner cabin. These works have led to an in-depth understanding of the physics with respect to the airflow fields in cabin environments, which has laid a foundation on which further investigations on contaminant transport could be conducted. However, their studies were mainly focused on the airflow field in the first-class cabin where the occupant density is much lower than that in an economy-class cabin, while the transport of particulate matters was not included. The effect of passenger body heat in the first-class cabin may be not an appropriate representation of that in an economy-class cabin due to the lower occupant density. Considering that passengers in an economy-class cabin may be subject to a higher risk associated with particulate exposure, this study focuses on the transport and distribution characteristics of particulate matters in economy-class cabins.

The economy cabin section model was built based on a typical medium-size commercial airliner. For a realistic airliner cabin, the interior airflow field is very complicated due to the diversity of ventilation layouts and individual differences of passengers. However, according to Liu et al. (2012)’s experimental measurements, the airflow pattern is approximately symmetric across the central plane of the aisle (left-right). Also, the ventilation layouts were assumed to be uniform and the difference of transitional airflow pattern along the aisle (front-back) was excluded in this study. Therefore, a cabin section containing 3 seats and 3 passengers was built as the computational domain, as schematically illustrated in Figure 4.1. A symmetric boundary was applied on the side plane and translational periodicity boundaries were setup at the front and back surfaces of the cabin section, respectively. By applying the periodic boundary conditions, the exactly same air and particle information was specified at the front and back planes, which means when a particle leaves the domain from a periodic boundary, another particle enters the domain with the same velocity from the other periodic boundary.

A comparison of the economy-class cabin section model of this study with the first-class cabin model by Liu et al. (2012)’s was illustrated in Figure 4.2. The two cabin section models were very similar to each other while the seats model in this study had a larger overall width.
The ventilation inlet and outlet were located in the upper and lower sides of the cabin wall, close to the baggage compartments and the floor, respectively (Figure 4.1b). The ventilation rate was carefully set according to the ASHARE aviation standard (ASHRAE, 2009), which yielded an air mass flow rate of 0.04 kg/s at the inlet for the 3-passenger cabin section. The air temperature at the inlet was 25°C as recommended by the ASHRAE (2009). In order to capture the realistic airflow field in the vicinity of the passengers, a 3D-scanned female manikin model containing very detailed body features (available at www.ie.dtu.dk/manikin) was employed in this
study. Detailed information about the manikin geometry could be found in Sorensen and Voigt (2003). For heat transfer modelling, a fixed skin temperature of 31°C was applied at the manikin surface, which is consistent with the manikin skin temperature applied by Gao and Niu (2004). The skin temperature setting resulted in an equivalent convective heat load of 40 W, which is close to the manikin heat load (38 W) set by Topp et al. (2002). The other solid walls, including the cabin wall, the ceiling, the floor and the seats were assumed to be adiabatic.

![Figure 4.2: Comparison of the cabin model of this study (pink) and that by Liu et al (blue) (Liu et al., 2012) (This figure is not displayed here.)](image)

Particulate contaminants were assumed to be exhaled by the passengers through coughing. According to the experimental measurements by Redrow et al. (2011), the mean diameter of coughing droplets was about 13.5 microns, while most sputum droplets would evaporate and become droplet nuclei (3.5 microns in diameter) within 0.25s to 0.55s, depending on the relative humidity. Since the cabin space has a very low humidity (around RH 17% according to the spot measurements by Cui et al. (2014)), the droplets were expected to become droplet nuclei in a very short time. Therefore, the evaporation of sputum droplets was ignored and the droplet nuclei were treated as particles with constant diameter of 3.5 microns in this study.

Computations were conducted with particles released by each of the three passengers, respectively. The particles were tracked by using the Lagrangian approach,
which continuously tracks particle movement through the domain and is possible to model transient particle transport using a steady state simulation. By using this approach, the airflow field was steady state and thereby the transient cough cannot be considered in this study. According to our previous studies, the movement of particles larger than 10 m is dominated by the inertia forces (mainly from coughing or sneezing) and gravity, whilst the transport of particles smaller than 10 m is dominated by the airflow (Li et al., 2012, 2013). As the particle size in this study was 3.5 m, the particles will be quickly carried on by the local airflow from the ventilation system. Therefore, the impact of cough release on the local airflow was not considered.

The cabin geometry model as illustrated in Figure 4.2b was discretised using unstructured mesh by ICEM 14.5. For the purpose of achieving accurate captures of the body features and airflow field in the vicinity of the manikins, very fine meshes were built around the manikin skin, whilst relative coarse mesh were applied at less significant surfaces such as cabin walls and seats. Grid sensitivity study was performed to test the mesh quality and mesh independency was achieved at 1.5 million mesh elements.

4.2.2 Mathematical Models

The airflow field was solved using the incompressible Navier-Stokes equations incorporated with the Buossinesq approximation accounting for the thermal buoyancy flow induced by the passenger body heat. Thus, the thermal expansion of air was calculated in terms of a reference density $\rho_{\text{ref}}$ by:

$$\rho_a = \rho_{\text{ref}} (1 - \beta(T_a - T_{\text{ref}}))$$

where, $\beta$ is the thermal expansion coefficient and $T_{\text{ref}}$ is the reference temperature corresponding to $\rho_{\text{ref}}$.

The particles were tracked using the Lagrangian approach. For micron particles with a diameter of $d_p$ immersed in continuous air, important forces governing particle motions are the drag force $\vec{F}_D$ and the buoyancy force $\vec{F}_{\text{Buoy}}$ (Li et al., 2012), thus,

$$m_p \frac{d\vec{U}_p}{dt} = \vec{F}_D + \vec{F}_{\text{Buoy}}$$

$$\vec{F}_D = \frac{C_D \pi d_p^2}{2} \rho_g |\vec{U}_p - \vec{U}| (\vec{U}_p - \vec{U})$$

46
According to the experimental measurements by Liu et al. (2012), the airflow in an airliner cabin is typically low in velocity and high in turbulence. As the fluctuating component of the air velocity is the main source that causes the dispersion of aerosol particles, the effect of turbulent dispersion on particle transport is modeled by adding an eddy fluctuating component onto the mean air velocity when a particle enters into the eddy. Therefore, the local air velocity is redefined by

\[
\vec{U} = \bar{U} + \Gamma \left( \frac{2k}{3} \right)^{0.5}
\] (4.5)

where, \( \Gamma \) is a normally distributed random number which accounts for the randomness of turbulence about a mean value.

Each eddy has its unique fluctuating velocity \( U' \), lifetime \( t_e \) and length \( l_e \). The fluctuating eddy velocity was added only when the interaction time between particle and eddy time is less than the eddy lifetime and the displacement of the particle relative to the eddy is less than the eddy length. Conversely, the particle is assumed to be entering a new eddy with new fluctuating velocity, lifetime and length.

### 4.3 Results and Discussion

The governing equations were solved using the commercial CFD software CFX 14.5. Due to its successful application in modeling indoor air, the RNG k-\( \varepsilon \) model was employed in this study for the air turbulence with a high resolution advection scheme. Sensitivity tests were also performed for the number of representative droplet trajectories tracked by the Lagrangian approach. It was found that the numerical results became stable when the number of droplet trajectories increased up to 20,000. Convergence was achieved when the RMS residual of the continuity equation decreased to be lower than \( 1 \times 10^{-6} \). For the purpose of comparison, computations were also conducted with the isothermal condition where heat transfer was excluded from the model.

#### 4.3.1 Fluid Flow

Airflow fields yielded from both the isothermal and thermal conditions were presented and compared in Figure 4.3, where Plane 1, 2 and 3 are the planes crossing the
manikin bodies, 20 mm in front of the manikin nose tips, and crossing the manikin legs, respectively.

Figure 4.3: Velocity vector field across Y-plane for isothermal and thermal case.

Figure 4.3 revealed that the overall airflow in the cabin was mainly driven by the ventilation jet, while the thermal effect would completely change the overall airflow pattern. When heat transfer was excluded from the model, all these three selected planes were showing very similar airflow patterns that a large counter-clockwise vortex was obviously observed in the central bulk region of each plane while a small clockwise vortex appeared in the upper part of the aisle region close to the cabin roof and baggage compartments (Figure 4.3a-c). The airflow pattern was dominated by the inlet air jet as well as the geometry of manikins and seats. However, with the inclusion of heat transfer into the model, the buoyancy driven thermal plume dramatically changed the airflow pattern. At first, the size of the large vortex in the bulk region was largely suppressed and the small vortex close to the cabin roof was nearly eliminated in
all the 3 selected planes. Secondly, significant uprising buoyancy flows were observed above the passenger shoulders, this especially true for the region between passenger A and B (Figure 4.3d). However, strong uprising buoyancy flow was only observed in Plane 1, which indicated that the impact of thermal effect became less significant with increased distance from the manikin torso as the heat would be quickly dispersed after leaving the manikin bodies.

The predicted airflow field in Plane 2 under the isothermal condition (Figure 4.3b) was compared against the PIV experimental results by Liu et al. (2012), as illustrated in Figure 4.4. The MD-82 airliner cabin in the experiments had a similar shape with the cabin model in this study, but had different sizes, therefore, the Y and Z coordinates were normalized in Figure 4.4 for the convenience of comparison. Figure 4.4 demonstrated that the overall airflow patterns yielded from the experimental measurements (Liu et al., 2012) and numerical predictions were very close. The large counter-clockwise vortex almost occupying the whole bulk region was observed in both experimental measurements and the numerical prediction. However, velocity magnitude difference and direction variations can be noticed at some region between these two experimental data and numerical results. These differences were mainly caused by the geometry difference between the test cabin and the computational model.

![Figure 4.4: Comparison of airflow vectors between experimental data and computational results.](image)

49
The local airflow fields in the passengers’ breathing zones were illustrated in Figure 4.5. When heat transfer was excluded from the models, the breathing zone was completely controlled by the ventilation jet and cabin and manikin geometry. As shown in Figure 4.5a-c, an uprising airflow existed in front of Passenger A who was seated close to the windows, a descending airflow was observed in the breathing zone of Passenger C who was next to the aisle while Passenger B was located in the middle of vortex (see Figure 4.3b). However, when heat transfer was included in the models (Figure 4.5d-f), significant uprising airflow appeared in the vicinity of the manikin surface. For Passengers A and B, stronger uprising airflows were observed in their breathing zones, while for Passenger C the descending airflow in the breathing zone (Figure 4.5c) was completely replaced by the uprising buoyancy flow (Figure 4.5f).

Figure 4.5: Velocity vector field across X-plane for isothermal and thermal cases.
Figure 4.6: Comparison of vertical velocities between experimental and computational results along selected lines.
The predicted velocity distributions along vertical lines in front of the manikins were extracted and compared against Liu et al. (2012)’s experimental data, as illustrated in Figure 4.6. Velocity profiles along totally 6 vertical lines were analysed and compared. The locations of the vertical lines were also included in the figure. Figure 4.6 revealed that the overall agreement between the numerical results and the experimental data was satisfactory. Especially for Line 1, 2 and 3, the CFD models realized an accurate capture of the air velocity profiles. Comparatively, the agreement for Line 4, 5 and 6 was not so good, this was perhaps due to the different geometries between the test cabin (Liu et al., 2012) and the computational domain of this study, as illustrated in Figure 4.2. Liu et al. (2012) measured the first-class cabin with only 2 seats in a section while 3 seats were arranged in the same section. However, this difference did not cause much change in the airflow field represented by Line 1, 2 and 3 because this region was constantly affected by the passengers and less affected by the geometry difference. Comparatively, the air velocity profiles along Line 4 - 6 were significantly changed due to the different geometries.

The numerical results revealed that the airflow field in the vicinity of a manikin was highly sensitive to the position. For the purpose of quantitative analysis and comparison, the velocity profiles along 3 selected vertical lines were also plotted and compared, as illustrated in Figure 4.7. The positions of the vertical lines relative to a certain manikin were kept constant, where Line 1 (marked as Position 1 in Figure 4.7) was located above the manikin head, Line 2 (marked as Position 2 in Figure 4.7) was immediately in front the manikin face and Line 3 (marked as Position 3 in Figure 4.7) penetrated through the middle point between the manikin knees.

Figure 4.7 revealed that the airflow field in the upper part of the cabin was significantly affected by the inlet jet. At Position 1, the velocity magnitude above Passenger A was the highest while that above Passenger C was the lowest as Passenger A was seated close to the ventilation jet while Passenger C was farthest from the jet. The ventilation jet dispersed soon after entering the cabin. At Position 2, the air velocity profiles in the upper part were still affected by the ventilation jet, while this effect was not significant in the low height. The air velocities were fluctuating in the vicinity of manikin torsos and started increasing above manikin bodies due to the buoyancy driven thermal plume. Despite the fluctuation, the airflow velocity near the torsos was quite uniform for different passengers.
4.3.2 Droplets Transport and Distribution

The transport and distribution characteristics of droplets generated by the coughs from the passengers were studies under both isothermal and thermal conditions. Separate computations were conducted to simulate the tracks of the droplets exhaled by each of the three manikins, as illustrated in Figure 4.8. It is clear that the passenger body heat not only had a significant effect on the airflow field as illustrated in Figure 4.3 and 4.4, it also dramatically altered the trajectories of droplet transport. Figure 4.8 also demonstrated that the droplet trajectories were jointly controlled by the ventilating airflow and the buoyancy driven thermal plume. The droplets would be elevated to a higher level, descend towards the floor, or even be locked up in the breathing zone, depending on the intensity of the thermal plume.
Figure 4.8: Droplets trajectories for isotheral and thermal cases.

For Passenger A (Figure 4.8a and d) who was seated close to the windows, the exhaled droplets were travelling with very limited distance rather than travelling widely through the entire cabin section. Gradually, the droplets formed a lock-up circle in front of himself, especially at the breathing zone, in both isothermal and thermal cases. However, when heat transfer was included in the model, the droplets lock-up was more intense and the lock-up area was wider due to the balance of the uprising thermal plume against the descending ventilation airflow. Figure 4.8b and e illustrated the trajectories of droplets exhaled by passenger B who was sitting in the middle of the seats. It was noticed that when heat transfer was excluded, the model predicted that the droplets exhaled by Passenger B were mainly concentrated in the bulk air above the passengers (Figure 4.8b), thus Passenger A was located in a region without droplets. However, when heat transfer was included in the model, the droplets were predicted to be distributed in a wider range (Figure 4.8e). The transport trajectories of droplets exhaled by Passenger C were illustrated in Figure 4.8c and f. As Passenger C was located in the region affected by the inlet jet, the
thermal effects of body heat was not so significantly detectable. After leaving the release point (Passenger C), the droplets quickly dispersed and joined the overall flow vortex and travelling widely in the cabin.

An overall review of Figure 4.8 demonstrated that as heat transfer was included in the model, obvious droplets ascending was observed immediately after the droplets were exhaled by the passengers seated outside the inlet jet (e.g., Passenger A and B), which caused significant changes in the droplet trajectories from those yielded from the isothermal cases. However, when location of droplet release was located in the inlet jet (Passenger C), the thermal effect was largely suppressed by the inlet jet. It was clear the transport characteristics of droplet transport were largely affected by the airflow field as represented by the ascending movement of the droplets induced by the uprising thermal plume. However, a comparison of Figure 4.8 against Figure 4.3 demonstrated that the spatial distribution of the droplets in the cabin has strong local characteristics, and thus could not be fully represented by the airflow field.

4.3.3 Droplets Transport and Assessment of Health Risks

In order to further investigate the transient characteristics of droplet transport in the cabin environment under the thermal condition, the transient droplet distributions at $t = 5s$, 30s and 120s were analysed, as illustrated in Figure 4.9. It demonstrated that dispersion speed of droplets through the air the range they could reach were highly sensitive to their location of release. When the droplets were released by Passenger A (Figure 4.9a, they move upwards slowly with the airflow, then suddenly changed their way downwards when they reached the inlet jet ($t = 5s$). Then the droplets were carried on by the airflow. At $t = 30s$, the droplets were fully dispersed in front of Passenger A. The vortex flow as illustrated in Figure 4.3f then locked the droplets for a quite long time. Until $t = 120s$, there were still a number of droplets presented in front of Passenger A. Comparatively, the dispersion speed of the droplets released by Passenger B is much higher. At $t = 5s$, the droplets had been transported in the bulk region while at $t = 30s$, the droplets were fully dispersed in the whole cabin section. At $t = 120s$, there were just a few droplets remained in the breathing zone. When the droplets were released by Passenger C who was affected by the ventilation jet, the exhaled droplets mainly affect Passenger B and C, while Passenger A was not significantly affected.
Figure 4.9: Droplets transport and distribution at various time steps.
According to Figure 4.9, under a given airflow field, the transport and distribution characteristics of exhaled droplets were highly sensitive to the location of release. Therefore, the health risk impacted by the droplets released by a sick passenger on other passengers may be different. In order to achieve a quantitative result of exposure risk assessment, the droplet residence time in each manikin’s breathing zone was analysed. According to the Australia Work Safety Standard (Australia, 2013), the personal breathing zone of a person was defined as a hemisphere of 300 mm radius extending in front of the face and measured from the midpoint of an imaginary line joining the ears. The averaged residence time of the droplets exhaled by a passenger in other passengers’ breathing zone was illustrated in Figure 4.10. It was clear that the droplet residence time is sensitive to both the location of release and the interested breathing zone. According to Figure 4.10, the droplets exhaled by Passenger A had longer residence time that those exhaled by Passenger B and C. The longest droplet residence time appeared in Passenger C’s breathing zone when the droplets were released by Passenger A, which was around 6.5s. Comparatively, the droplet released by Passenger B and C are more easily to be carried on by the airflow. The computations also revealed the residence time of the droplets released by Passenger A in his own breathing zone could be as long as 9.5s. Such a long droplet residence time indicated a poor ventilation in Passenger A’s breathing zone, therefore, in order to minimize the risks thus caused, the personalized ventilation was strongly recommended for Passenger A.

![Figure 4.10: Average droplet residence time in the passengers’ breathing zones.](image-url)
4.4 Conclusion

This study applied a section of airliner cabin containing 3 seats and 3 passengers to investigate the thermal effects of passenger body heat on the airflow field and the transport characteristics of exhaled droplets. The simulations were conducted with isothermal and thermal conditions and the numerical results were validated using experimental data and compared against each other. Conclusions arising from this study are as follows:

1. The thermal buoyancy flow driven by the passenger body heat has a significant effect on the overall and local airflow fields in the cabin section. The thermal plume effect was maximised in some local regions, e.g. in front of passengers, between two passenger shoulders and above passenger heads under typical cabin environment. The intensity of thermal plume may vary among different passengers, depending on the sitting locations of passengers and cabin geometry.

2. The transport and distribution characteristics of the droplets exhaled by the passengers were highly sensitive to the location of release. When droplets were released by the passenger close to the window (Passenger A), they may have longer residence time in other passengers’ breathing zones.
The main findings of this chapter have been published in:


This paper presented a comparative study of various mathematical models (including a new type of Eulerian-based two-phase flow model, known as the Eulerian-Eulerian model) for modelling the PM transport and distribution characteristics in indoor spaces. Computations were conducted with both transient and steady states under isothermal and thermal conditions. Comparisons against the experimental data available in the literature and the existing models for PM transport (e.g. the Lagrangian model and the drift-flux model) demonstrated that the Eulerian-Eulerian model is capable of realizing a comprehensive description of inter-phase mechanisms and has a comparable accuracy with the Lagrangian model. More importantly, the Eulerian-Eulerian model gives a direct prediction to the PM concentration field through solving a set of conservation equations for the particulate phase, thus does not need additional post-processing procedures to estimate the PM concentration based on the particle trajectories. Therefore, the E-E model needs much lower computational cost than the Lagrangian model and eliminates the uncertainties that might be caused by the additional procedures.
5.1 Introduction

As a cost-efficient predictive approach, computational fluid dynamics (CFD) has been widely employed in investigating particulate matters (PMs) in indoor environments (Ahmadi, 2012) and assessing health risks associated with the exposure to PMs (Inthavong et al., 2013). It is well known that CFD simulations are based on appropriate fluid dynamics equations. Therefore, reliable two-phase flow models are indispensable and have to be incorporated in the CFD models when simulating PM transport. During the past years, various two-phase flow models such as the Lagrangian model (Zhang and Chen, 2007b; Zhao et al., 2008) and the simplified Eulerian models including the mixture model (Zhao et al., 2008) and the drift-flux model (Zhao et al., 2008; Chen et al., 2006) have been used to model PM transport in indoor spaces.

The Lagrangian approach, which tracks a number of representative particles separately through the air, is the most popular two-phase flow model for modelling PM transport and has its unique advantage in whole-process tracking of particle movement from the injection point to the final destination, and vice versa. It also allows an integrated inter-phase coupling which could include various interacting mechanisms between the phases. For example, in most studies employing the Lagrangian model (Zhang and Chen, 2007b; Zhao et al., 2008), the drag force, the buoyancy force and additional forces were effectively incorporated so that a complete description of the forces acting on the particles was achieved. However, as only the motion equation is solved for the particles, the Lagrangian approach cannot give a direct prediction to the PM concentration, which is actually of more significant practical importance than the particle trajectories especially in health risk assessments as the PM concentration was found to be strongly associated with morbidity and mortality (Pope et al., 1995). Therefore, additional post-processes are requested to convert the particle trajectories into the PM concentration. During the past years, the sampling volume method (Zhang and Chen, 2007b; Salmanzadeh et al., 2012) and the kernel method (Chang et al., 2012b) have been developed to estimate the particle concentration based on the particle trajectories. However, these post-process procedures are still suffering from poor stability and inaccuracy, as reported by Chang et al. (2012b,a).

On the other hand, as the PM concentration in general indoor spaces is quite low (lower than 25 μg/m³ for PM2.5 in terms of the WHO guidelines (WHO, 2005) and the particle size is generally small (micron or sub-micron), it’s safe to assume that the existence of PMs in the air has no detectable effects on the airflow field. This
has made a couple of simplified Eulerian two-phase flow models such as the mixture model (Zhao et al., 2008) and the drift-flux model (Zhao et al., 2008; Chen and Chen, 2010) applicable to PM transport in indoor air. In the mixture model, the air-PM mixture is treated as a pseudo fluid whose properties are calculated based on the local volume fractions of the two phases. In the drift-flux model, the PM concentration is numerically treated as a transportable scalar and a conservation equation is solved for the scalar. Although these simplified Eulerian models could give a direction prediction to the PM concentration, it’s hard to realize a complete incorporation of inter-phase mechanisms into the models. For example, in the mixture model by Zhao et al. (2008) only the inter-phase slip velocity was empirically considered in terms of the drag force, while in the drift-flux model by Zhao et al. (2008) and Zhang and Chen (2007b), only the particle settling velocity induced by the drag force and the buoyancy force was considered.

Zhang and Chen (2007b) and Zhao et al. (2008) compared these models in the aspects of predicting PM transport. They reported that (1) the Eulerian based models are much more cost-efficient than the Lagrangian model, and (2) the mixture model has the weakest capability of describing PM transport, the drift-flux model and the Lagrangian model have similar accuracy for steady state PM transport while the latter performs better for transient computations. It is supposed that the better performance of the Lagrangian model is attributed to its complete description of the inter-phase mechanisms. However, considering the drawbacks of the Lagrangian model in predicting the PM concentration, there naturally arises the question if there is a model which could take the best features of the Lagrangian model and the drift-flux model.

In this study, a new type of Eulerian-based two-phase flow model, known as the Eulerian-Eulerian model (Ishii, 1975; Ishii and Mishima, 1984), was introduced to model PM transport in indoor spaces. Being different from the Lagrangian model and the drift-flux model, the Eulerian-Eulerian model treats the dispersed PMs as a continuous phase inter-penetrating with the air and solves two sets of conservation equations governing the balance of mass, momentum and energy of each phase. Since the macroscopic fields of one phase are not independent of the other phase, interaction terms which couple the inter-phase transport of mass, momentum and energy appear in the field equations. This model is not only able to give a direct prediction to the concentration of the particulate phase (by volume or mass fraction) in a cost-efficient way, but also capable of realizing a complete description of the inter-phase
mechanisms, and is therefore expected to be promising in modelling various two-phase flows.

The Eulerian-Eulerian model has been widely employed and proven to be effective in modelling gas-particle flows with high particle loads such as those in fluidized beds (Altantzis et al., 2015; Cloete et al., 2015) and dust lifting devices (Utkilen et al., 2014; Ilea et al., 2008). Although modelling of PM transport in indoor environments using the Eulerian-Eulerian model has been rarely reported in the literature, the pioneering study by Armand et al. (1998), who applied the two-fluid model to simulate aerosol transport in isothermal laminar (the Re number as low as 100) and turbulent (the Re number up to 83,000) flows, has proven its validity for dilute gas-particle flows and its ability to calculate various complicated diphasic flows involving aerosols transport. In this study, the Eulerian-Eulerian model was further validated using the experimental data available in the literature and compared against the drift-flux model and the Lagrangian model in the aspects of computational cost, two-phase flow fields, PM concentration field and particle-wall interactions.

5.2 Mathematic Models

5.2.1 The Eulerian-Eulerian Model

The Eulerian-Eulerian model solves two sets of conservation equations, one for each phase. The conservation equations take the following form (ANSYS, 2015):

the continuity equation
\[
\frac{\partial}{\partial t} \left( \alpha_i \rho_i \right) + \nabla \cdot \left( \alpha_i \rho_i \vec{U}_i \right) = 0 \tag{5.1}
\]

the momentum equation
\[
\frac{\partial}{\partial t} \left( \alpha_i \rho_i \vec{U}_i \right) + \nabla \cdot \left( \alpha_i \left( \rho_i \vec{U}_i \vec{U}_i - \mu_i \left( \nabla \vec{U}_i + \left( \nabla \vec{U}_i \right)^T \right) \right) = \alpha_i \left( S_{Bouy} - \nabla P_i \right) + \vec{F}_{ij} \tag{5.2}
\]

where, \( i \) and \( j \) are the phase denotations (\( i, j = a \) for the air phase and \( i, j = p \) for the particle phase). \( \alpha \) is the volume fraction (\( \alpha_a = 1 - \alpha_p \)), \( \rho, \vec{U} \), and \( P \) represent the density, velocity and pressure, respectively.

The heat transfer within the particles and between the particulate phase and the air phase was ignored in this study. Therefore, the energy equation was solved only for the air phase
\[
\frac{\partial}{\partial t} \left( \alpha_a \rho_a H_a \right) + \nabla \cdot \left( \alpha_a \left( \rho_a \vec{U}_a H_a - \lambda_a \nabla T \right) \right) = 0 \tag{5.3}
\]
where, $H$ and $\Lambda$ are the enthalpy and thermal conductivity, respectively.

$S_{Buoy}$ is the momentum source due to buoyancy, which is defined in terms of a reference density $\rho_{ref}$.

$$S_{Buoy} = (\rho_i - \rho_{ref}) g$$  (5.4)

When modelling the inter-phase forces, the spherical particle assumption (ANSYS, 2015) was employed. For a spherical micron particle submerged in continuous fluid, the forces that may be significant include the drag force $\vec{F}_D$, the turbulent dispersion force $\vec{F}_{TD}$ and the virtual mass force $\vec{F}_{VM}$, which are defined by Equations 5.6-5.8, respectively.

$$\vec{F}_{ap} = -\vec{F}_{pa} = \vec{F}_D + \vec{F}_{TD} + \vec{F}_{VM}$$  (5.5)

$$\vec{F}_D = \frac{3}{4} C_D \frac{\alpha_p \rho_a}{d_p} \left| \vec{U}_p - \vec{U}_a \right| \left( \vec{U}_p - \vec{U}_a \right)$$  (5.6)

$$\vec{F}_{TD} = -C_{TD} \rho_a k_a \nabla \alpha_a$$  (5.7)

$$\vec{F}_{VM} = C_{VM} \alpha_p \rho_a \left( \frac{d\vec{U}_p}{dt} - \frac{d\vec{U}_a}{dt} \right)$$  (5.8)

Although the inter-phase forces are formulated in a mechanistic way, the coefficients (e.g. $C_D$, $C_{TD}$ and $C_{VM}$) are generally determined empirically. For the spherical particles, the turbulent dispersion coefficient was modelled according to Lopez de Bertodano Lopez de Bertodano (1991) and a constant $C_{VM} = 0.1$ was employed for the virtual mass coefficient ANSYS (2015). The drag coefficient $C_D$ is a function of the particle Reynolds number $Re_p$ and was modelled in this study using the Florin (1978) correlation, which is valid for $Re_p \leq 2000$.

$$C_D = \frac{24}{Re_p} \left( 1 + 0.149 Re_p^{0.687} \right)$$  (5.9)

### 5.2.2 Other PM Transport Models for Comparison

For the purpose of comparison, some other widely used PM transport models including the Lagrangian model and the drift-flux model were also included in this study. These models solve the same Eulerian equations (5.1-5.3) for the airflow field but use different approaches for PM transport.
The Lagrangian model uses the equation of motion to track the particle movement. The drag force \( F_D \), the buoyancy force \( F_{Buoy} \) and the virtual mass force \( F_{VM} \) are considered here in order to keep the same inter-phase momentum transfer mechanisms as those considered in the Eulerian-Eulerian model.

\[
m_p \frac{d\vec{U}_p}{dt} = F_D + F_{Buoy} + F_{VM}\tag{5.10}
\]

\[
F_D = \frac{C_D \pi d_p^2}{2} \rho_a \left| \vec{U}_p - \vec{U}_a \right| (\vec{U}_p - \vec{U}_a) \tag{5.11}
\]

\[
F_{Buoy} = \frac{\pi d_p^3}{6} (\rho_p - \rho_a) g \tag{5.12}
\]

\[
F_{VM} = \frac{C_{VM} \pi d_p^3}{2} \rho_a \left( \frac{d\vec{U}_p}{dt} - \frac{d\vec{U}_a}{dt} \right) \tag{5.13}
\]

Being different from the Eulerian-Eulerian model, the effect of turbulent dispersion on particle transport is modelled in the Lagrangian model by adding an eddy fluctuating component onto the mean air velocity. It is the fluctuating component of the air velocity that causes the dispersion of particles in turbulent flow. Therefore, the local air velocity is redefined by

\[
\vec{U}_a = \vec{U}_a + \Phi \left( \frac{2k}{3} \right)^{0.5} \tag{5.14}
\]

where, \( \Phi \) is a normally distributed random number which accounts for the randomness of turbulence about a mean value.

The drift-flux model treats the PM concentration as a transportable scalar. Several effects such as the inter-phase slip (Zhao et al., 2008) and particle diffusion (Chen et al., 2006) have been included in the model. The transport equation of the PM concentration takes the form of Equation 5.15.

\[
\frac{\partial}{\partial t} (\rho_a C_p) + \nabla \cdot \left( \rho_a C_p \left( \vec{U}_a + \vec{U}_s \right) - \Gamma_p \nabla C_p \right) = S_{Cp} \tag{5.15}
\]

where \( C_p \) is a scalar representing the PM concentration, \( \Gamma_p \) is the effective particle diffusivity and \( S_{Cp} \) is the source term. \( \vec{U}_s \) is the particle settling velocity defined by Zhao et al. (2008)

\[
|\vec{U}_s| = \sqrt{\frac{3 g d_p \rho_p - \rho_a}{4 f_D \rho_a}} \tag{5.16}
\]
Obviously, all the three models solve the same Eulerian equations for the airflow field. The difference between them lies in the approaches they utilize to deal with the particulate phase. Therefore, for the purpose of a constant nomenclature, the above models are abbreviated in the following sections as the E-E model for the Eulerian-Eulerian model, the E-L model for the Lagrangian model and the E-S (S stands for Scalar) model for the drift-flux model, respectively.

5.3 Numerical Procedures

The experimental data by Chen et al. (2006) were utilized in this study for model validation and comparison. In their experiments, a ventilated chamber with dimensions of 0.4m-width × 0.4m-height × 0.8m-length was used for the measurements. Air at the room temperature together with particles with an average diameter of 10µm and a density of 1400kg/m³ was injected from the inlet with a mean velocity of 0.225m/s and then exhausted via the outlet located on the other end of the chamber. Both the inlet and outlet have the dimensions of 0.04m × 0.04m. The airflow velocity and particle concentration data were obtained using a phase Doppler anemometry (PDA) system.

Figure 5.1: The computational domain based on Chen et al. (2006)’s experimental setup.
Due to the symmetric distribution of the two-phase flow field about the central plane, a half chamber was built as the computational domain, as illustrated in Figure 5.1. The domain was then discretized using structured hexahedral mesh. Mesh sensitivity test proved that when the mesh density exceeded $40 \times 80 \times 160$ cells, mesh independence was achieved for all the three models since a further increase of the cell number up to $50 \times 100 \times 200$ just caused a small change of less than 0.1% in the predicted particle concentration.

During the computations, uniform velocity and concentration profiles were specified at the inlet and an atmospheric pressure boundary condition was applied at the outlet. Particle-wall interaction is another important boundary condition deserves careful formulating. In the E-E model, since the PMs were treated as a continuous phase, it was hard to describe the realistic particle behaviours such as deposition, collision and bouncing. As a simplification, particle deposition was ignored and a free-slip boundary condition was applied at the chamber walls for the particulate phase. In order to achieve a similar modelling of the particle-wall interactions, particle deposition was also ignored in the E-L model and a full bouncing boundary condition was applied at the walls, which assumes that the particles bounce back with the same magnitude of momentum after hitting the chamber walls. Similarly in the E-S model, particle deposition was excluded as well and the PM transport in the near-wall regions was simply controlled by the particle diffusion.

As indicated by Equation 5.5, the inter-phase forces are naturally fully-coupled for the both phases in the E-E model. In order to achieve a similar description of the inter-phase forces, the "two-way coupling" algorithm (ANSYS, 2015) was employed for the E-L model, which realizes a two-way momentum transfer between the phases. Unfortunately, the effects of particles on the airflow field could not be considered in the E-S model due to the inherent limitation.

Since the flow was isothermal in the experiment (Chen et al., 2006), the energy equation (Equation 5.3) was not solved for this case. The model equations were discretized using the finite volume method and then solved using the commercial CFD code ANSYS CFX 14.5. During the computations, a steady-state airflow field was firstly obtained, and subsequently a transient simulation was performed for the particulate phase based on the airflow field. The total particle tracking time was 1800 s, which was sufficiently long to observe the particle dynamic behaviours in a micro-environment (Chen et al., 2006).

Different approaches of modelling PM transport led to different forms of presenting the numerical results. The E-E model predicts the PM volume fraction which is
a real sense of concentration, the E-S model gives a scalar representing the PM concentration while the E-L model yields particle trajectories. In order to keep the comparability between the numerical results, the particle trajectories yielded from the E-L model needed to be converted to the PM concentration. In this study, the so-called particle source in cell (PSI-C) method developed by Zhang and Chen (2007b) was utilized to calculate the PM concentration. To apply the PSI-C method, the test chamber containing the PM trajectories was firstly discretized using a number of control volumes (cells). Then, the local PM concentration in a given cell could be estimated based on the particle residence time by

\[ C_j = \frac{M \sum_{i=1}^{m} dt(i,j)}{V_j} \]  

(5.17)

where \( C_j \) is the local PM concentration in the \( j^{th} \) cell and \( V_j \) is the volume of that cell, \( M \) is the mass flow rate represented by a particle trajectory and \( dt(i,j) \) is the residence time of the \( i^{th} \) particle in the \( j^{th} \) cell. It should be noted that the control volumes here for PM concentration calculation are different from the computational cells for model solution.

Similarly, the local PM velocity was calculated based on the mean velocity of the particle trajectories in that cell.

\[ u_{p,j} = \frac{\sum_{i=1}^{m} u_{p,i}}{m} \]  

(5.18)

5.4 Results and Discussion

5.4.1 Model Validation and Comparison

The turbulence model was carefully selected. Generally, the RNG k-\( \varepsilon \) model is thought to be more proper for indoor airflows than other turbulence models and has been widely employed in CFD simulations of indoor contaminant transport (Li et al., 2013; Liu et al., 2013). This is especially true for indoor airflows with thermal buoyancy flows (Li et al., 2013). However, for the isothermal flow as shown in Figure 5.1, things may be different. Both the standard k- model and RNG k- model were tested in this study before the PM transport simulations. The predicted V component profiles (in the Y direction) of the air velocity along the three vertical lines (Line 1, Y=0.2 m, Line 2, Y=0.4m and Line 3, Y=0.6m, see Figure 5.1) in the symmetric plane (\( X = 0m \)) were compared against the experimental data in Figure 5.2. It proved that the standard k- model achieved a better prediction of the airflow.
field than the RNG k-ε model. Especially in the regions with higher air velocity, e.g., the 0.3m < Z < 0.4m region affected by the inlet jet, the k-ε model performs much better than the RNG k-ε model. This was perhaps due to the small size of the test chamber and the flow in it was more like an in-duct flow which made the standard k-ε model more applicable. Therefore, the standard k-ε model was selected for the air turbulence in the following computations with the model shown in Figure 5.1.

Figure 5.2: Air velocity component profiles predicted with different turbulence models.

The CFD computations firstly proved that the inlet PM concentration had a significant effect on the predicted concentration and velocity fields of the particulate phase in the chamber. Unfortunately, the particle injection concentration at the inlet during the experiments was not given by Chen et al. (2006). Therefore, different inlet concentrations varying from 20 µg/m³ to 1000 mg/m³ were tested. The lower limit of the inlet concentration (20 µg/m³) was based on the WHO guidelines for the recommended annual mean concentration of PM10 (WHO, 2005) as good air quality. Then the inlet concentration was gradually increased until an acceptable agreement with experimental data was achieved. Figure 5.3 illustrated the PM concentration profiles alone Line 3 predicted by the E-E model with different inlet concentrations. For the convenience of comparison, the PM concentrations were normalized based on the corresponding inlet concentrations. Figure 5.3 demonstrated that when the
inlet concentration was 800\(mg/m^3\), the numerical results agreed well with the experimental data. This inlet concentration was believed to be quite reasonable for a PDA measurement when one considers the operability.

![Graph showing concentration profile](image)

**Figure 5.3:** Effect of inlet concentration on the concentration profile alone Line 3 (10\(nm\), \(t = 1800s\)).

With the inlet concentration of 800\(mg/m^3\), the predicted velocity fields of the particulate phase in the symmetric plane (\(X = 0m\)) were illustrated in Figure 5.4. Since the E-S model did not solve a transport or motion equation for the particulate phase, it could not predict the PM velocity. Therefore, the E-E model was compared only against the E-L model in terms of the PM velocity field. Figure 5.4a and b revealed that the E-E model and E-L model gave very close predictions to the PM velocity field. The both models generated a spreading and decelerating jet from the inlet. The overall airflow patterns were very similar except for some minor deviations in few local areas. Both the low PM velocity region in the upper-Y and upper-Z corner and the high PM velocity region near the outlet were successfully predicted. A quantitative comparison of the PM velocity profiles along the three vertical lines (Figure 5.4c) further proved that the both models have very close accuracy in terms of the PM velocity prediction.
Figure 5.4: PM velocity fields predicted by the E-E model and the E-L model (10\(\mu\)m, 800\(mg/m^3\), 1800s)
However, Figure 5.4 also revealed that the PM velocity contour curves yielded from the E-E model were smooth while the E-L model generated coarse contours. The fluctuation in the PM velocity profiles predicted by the E-L model was as high as 50% in some local regions (see Figure 5.4c for an example). In fact, the E-L model solved the PM motion by tracking the particles individually through the air, which yielded a discontinuous computational results represented by the particle trajectories. In order to keep the comparability of the predicted results by the models, the discontinuous particle trajectories yielded from the E-L model had to be converted into a continuous velocity field using the PSI-C method (Zhang and Chen, 2007b). Although an approximately continuous velocity field was obtained as shown in Figure 5.4, it still contained the inherent characteristics of the discontinuous trajectories. The discontinuous characteristics could also be observed in the concentration contours in the following sections.

Figure 5.5: PM concentration predictions vs. experimental data (10\(\mu\)m, 800mg/m\(^3\), 1800 s).

Figure 5.5 illustrated the comparison of the normalized PM concentration profiles along the vertical lines at \(t = 1800s\) against the experimental data of Chen et al. (2006). According to Figure 5.5, all of the three models achieved satisfactory agreement with the experimental data in the bulk region. However, the E-E model and the
E-L model predicted that a thin but still remarkable high-concentration region existed close to the bottom surface of the chamber while the E-S model failed to predict this high-concentration region. Although the experimental data of PM concentration in the regions immediately next to the chamber bottom were not available, it was still clear according to Figure 5.5 that there was a slight increase in the PM concentration when the height (Z) approached to zero. However, the E-S model predicted that the PM concentration decreased with descending height in the regions close to the chamber bottom, which conflicted with the experimental observations. On the contrary, the E-E model and the E-L model gave closer predictions in these regions.

For a clearer view of both the PM concentration filed in the bulk region and the high-concentration region close to the chamber bottom, the PM concentration contours in the $X = 0$ plane were also given, as shown Figure 5.6. It indicates that the models had similar predicting ability for the bulk region while E-E model and E-L model predicted a higher particle concentration near the chamber bottom.

Figure 5.6 also demonstrated the thin region of high PM concentration was so large that it almost covered the whole bottom surface of the chamber. The PM concentration in this thin region was even higher than the inlet concentration, which indicated particle accumulation occurring near the bottom surface. In fact, the computations demonstrated that for a transient transport process of particles as large as 10 $\mu$m, particle accumulation could always be observed near the chamber bottom, even for very low inlet concentrations. Figure 5.7 illustrated the distribution of particles with an inlet concentration of $487 \mu g/m^3$, predicted by the E-L model. It could be seen that the particles near the bottom had very low velocity (lower than 0.01 m/s, also see Figure 5.4c), which made the gravity settling a predominant effect on particle motion so that particle accumulation was observed at the very early stage ($t = 120s$). Considering that a fully-bounce boundary condition was applied at the chamber walls, the E-L model of this study could not capture particle deposition, however, it clearly predicted the regions in which significant particle deposition could happen. Similarly, by using a free-slip boundary condition for the particulate phase, the E-E model also predicted the particle accumulation near the chamber bottom which could be used to help identifying the regions with significant particle deposition. It also can be expected that by incorporating appropriate boundary conditions for the particle-wall interactions, both the E-E model and the E-L model are hopeful to be capable of predicting realistic particle behaviours such as deposition, collision and bouncing. On the contrary, although the E-S model considered gravitational
settling of the particles through an empirical settling velocity, it failed to predict the particle accumulation near the chamber bottom.

Figure 5.6: PM concentration contours at t = 1800 s (10\(\mu\)m, 800mg/m\(^3\), 1800s)
5.4.2 PM Transport Prediction

It should be noted that the model validation was conducted with a quite high inlet PM concentration (800 mg/m$^3$), which was much higher than the acceptable PM concentrations as recommended by the WHO guidelines (10 µg/m$^3$ annual mean, 25 µg/m$^3$ 24-hour mean for PM2.5 and 20 µg/m$^3$ annual mean, 50 µg/m$^3$ 24-hour mean for PM10) (WHO, 2005). Even, according to a survey by Wang et al. (2013), the peak concentration of PM2.5 in Beijing (China), one of the worst polluted cities in the world, was 487 µg/m$^3$ for PM2.5 in the year of 2011 with an annual mean value of 98.85 µg/m$^3$, which was also far lower than the PM concentration utilized in the validation. As was demonstrated before, since the PM transport and distribution characteristics were affected by the inlet concentration, it’s therefore important to analyse the models with more realistic boundary conditions.

Figure 5.3 also demonstrated that when the inlet concentration was sufficiently low (< 1 mg/m$^3$), the predicted PM concentration fields were free from the effect of inlet concentration. Meanwhile, according to Abdullahi et al. (2013), who conducted a survey of the PM concentrations in over 100 kitchens from 12 countries in Asia, Europe and North America, more than 90% of these kitchens had a concentration lower than 1.0 mg/m$^3$. As kitchens are generally thought to have higher PM concentrations than other indoor spaces, thus, for most indoor spaces, the distribution pattern of PM concentration is expected not to be affected by the concentration value itself. Therefore, an inlet concentration of 487 µg/m$^3$ representing the worst PM pollution in Beijing Wang et al. (2013) was employed. In fact, the computations demonstrated
that a variation in the inlet concentration within the range of $< 1.0 \text{mg/m}^3$ didn’t cause any visible change in the normalized concentration pattern.

Figure 5.8: Predictions of PM concentration evolution (by the E-E and the E-L models, 487$\mu\text{g/m}^3$, 10$\mu\text{m}$).

The transient process of the development of the normalized PM concentration field in the $X = 0$ plane was illustrated in Figure 5.8. With a particle size of 10$\mu\text{m}$, significant particle accumulation close to the chamber bottom was predicted by the E-E model and the E-L model while was not predicted by the E-S model, therefore, the E-E model was compared only against the E-S model. Figure 5.8 illustrates that after the particles entered the chamber with the inlet air jet, they dispersed
as proceeding ahead \((t = 60s)\). After they hit the other end of the chamber, they bent their way downwards. Some of the particles then escaped through the outlet and some of them bent their way again towards the lower \(Y\) wall and then towards the upper \(Z\) wall. This pattern of particle movement thus caused a loop-like region with higher PM concentration \((t = 300s)\). Inside the loop was a region with lower PM concentration. Then, as the loop was expanding its size, the PM concentration in it was also increased, which led to an increasingly even PM concentration field \((t = 600s)\). At \(t = 900s\), no obvious concentration gradient could be observed in the chamber. This was significantly different from the concentration fields illustrated in Figure 5.6 where the PM concentration was high \((800mg/m^3)\).

Figure 5.8 also demonstrated that even for a low PM concentration, the E-E model achieved a comparable accuracy with the E-L model. The PM concentration fields predicted by the both models agreed very well with each other, for both the bulk region and the region close to the chamber bottom.

![Figure 5.8](image)

Figure 5.8: PM concentration evolution predicted by the E-E model (487\(\mu g/m^3\), 0.2 \(\mu m\)).

Computations were also conducted with ultrafine particles. Figure 5.9 illustrated the PM concentration fields predicted by the E-E model with the particle diameter of 76
0.2\(\mu m\). Compared with the coarse particles (10\(\mu m\), Figure 5.8), although the concentration field followed a similar revolving process, the dispersion of ultrafine particles in the inlet air jet was not so remarkable and their motion was predominantly controlled by the airflow, so that a significant high concentration region was observed in the inlet jet. Furthermore, particle accumulation close to the chamber was not observed with the ultrafine particles. This was possibly because that ultrafine particles are easily suspended in the air and are hard to depose.

### 5.4.3 Computational Cost

The computations were performed on a HP Z600 workstation with 8 processors and 24GB RAM. For each computational case of transient particle transport (the total particle tracking time was 1800s) using different two-phase flow models, the average computational times were compared in Table 1. The comparison proved that the E-S model had the lowest solution cost since it solves only one transport equation for the scalar representing the PM concentration. The E-E model had relatively higher solution cost since both the continuity equation and the momentum equation were solved for the particulate phase. Due to the individual particle tracking algorithm, the E-L model requested the highest solution cost. This was consistent with the conclusion drawn by Zhang and Chen (2007b) and Chen et al. (2006) that the Eulerian-based approach is much more computationally efficient compared to the Lagrangian approach.

However, the story did not stop here. When the PM concentration was requested, an additional post-processing procedure was needed to convert the Lagrangian particle trajectories into the concentration field. Unfortunately, most of the existing post-processing algorithms available in the literature are still suffering from poor numerical stability. As the PSI-C method (Zhang and Chen, 2007b) employed in this study was concerned, a number of different mesh structures and trajectory numbers had to be tested before a stable concentration could be finally obtained since the results could be very sensitive to the trajectory number and the size of the control volumes (Salmanzadeh et al., 2012). As demonstrated in Table 5.1, the time spent in calculating the PM concentration based on the particle trajectories was comparable with that for model solution. On the contrary, the E-E model and the E-L model could save a considerable computational cost by giving direct predictions to the particulate concentration.
Table 5.1: The computational costs by different models.

<table>
<thead>
<tr>
<th>Time cost (min)</th>
<th>E-E model</th>
<th>E-L model</th>
<th>E-S model</th>
</tr>
</thead>
<tbody>
<tr>
<td>Model solution</td>
<td>300</td>
<td>360</td>
<td>210</td>
</tr>
<tr>
<td>Post-processing</td>
<td>0</td>
<td>180</td>
<td>0</td>
</tr>
<tr>
<td>Total</td>
<td>300</td>
<td>540</td>
<td>210</td>
</tr>
</tbody>
</table>

5.5 Further Discussion

The studies by Armand et al. (1998) and Chen et al. (2006) were limited to isothermal conditions. However, heat transfer exists in most realistic indoor environments, which causes obvious thermal buoyancy flows and has significant effects on the characteristics of PM transport. Therefore, in order to further assess the E-E model for modelling PM transport in indoor environments, additional computations were conducted using a more realistic scenario.

![Figure 5.10: Model setup for computations with thermal condition.](image)

The computational model is illustrated in Figure 5.10, which contains a heat-releasing thermal manikin sitting in the middle of a room with displacement ventilation. The air exchange rate was $3h^{-1}$, which yielded an air supply rate of 0.066 $kg/s$ with a constant temperature of $18^\circ C$ at the low-momentum diffuser. The manikin model was from [http://www.ie.dtu.dk/manikin](http://www.ie.dtu.dk/manikin) (Sorensen and Voigt, 2003). Only convective heat loss from the manikin surface was considered and the total convective heat loss rate was 45W, as recommended by Rim and Novoselac (2009). Particles with a density of 1000 $kg/m^3$ and a diameter of 0.77 $\mu m$ were released from a circular
area (0.5\text{m} in diameter) located 0.5\text{m} upstream of the manikin so that the particle transport could be affected by the manikin thermal plume.

Steady-state computations were conducted using the E-E model and the E-L model, together with the RNG k-\varepsilon model for the air turbulence. When calculating the thermal buoyancy force, the Buossinesq approximation is employed in the momentum equation to account for the thermal expansion of air.

\[
\rho_a = \rho_{ref}(1 - \beta(T_a - T_{ref})) \tag{5.19}
\]

where, \(\beta\) is the thermal expansion coefficient of the air and \(\rho_{ref}\) is the reference density which takes value of the air density at the reference temperature \(T_{ref}\).

![Image](image1.png)

(a) The E-E model

![Image](image2.png)

(b) The E-L prediction by Salmanzadeh et al. (2012)

Figure 5.11: The predictions of thermal plume
For the thermal flow case, the E-E model and the E-L model generated very close airflow fields. A typical airflow field in the $Y = 0m$ plane is shown in Figure 5.11a, which indicates that a significant thermal buoyancy flow existed above the manikin head. The thermal buoyancy flow was so strong that it was the major airflow in the room, apart from the airflow near the floor which was driven by the ventilating jet. Salmanzadeh et al. (2012) also simulated the thermal plume of a seated manikin in a displacement ventilated room using the E-L model, their numerical results were included here for the purpose of comparison, as shown in 5.11b. Despite the different manikin geometry and room ventilation layout, the conditions in the simulations of Salmanzadeh et al. (2012) were very close to those in this study in the aspects of low momentum and buoyancy-driven flow. Figure 5.11 demonstrated that the thermal plumes yielded from the both models were phenomenologically analogous. The both simulations predicted that for a seated manikin in a low-momentum room, the thermal plume detached itself from the manikin from the back side of head, and then kept accelerating until reach the maximum velocity at around $0.40.5m$ above the head, which was consistent with the experimental observation by Marr et al. (2005). Figure 5.12 shows a quantitative comparison of the air velocity contours in a circular area ($0.25m$ in diameter) $0.25m$ above the manikin head. The both models gave very close predictions and the average air velocity in this area was $0.201m/s$ by the E-E model and $0.203m/s$ by the E-L model, with a deviation no larger than $1.0\%$.

![Air velocity contour in a circular area 0.25 m above the manikin head](image)

Figure 5.12: Air velocity contour in a circular area 0.25 m above the manikin head

Typical PM distributions around the manikin are illustrated in Figure 5.13a. It is clear that the E-E model (Figure 5.5a) gives a direct prediction to the particle concentration while the E-L model (Figure 5.5b) predicts the particle trajectories. Despite this, the overall PM transport or distribution patterns predicted by the models are
apparently very close. As the particles approached the manikin, they bent their way upwards due to the buoyancy effect of the thermal plume. After the particles hit the ceiling, they bent their way again into the horizontal direction, which caused the particles spreading all over the room. Locally, it’s important to notice that some particles which were released at a lower height were entrained into the breathing zone by the thermal plume. The particle concentration in the breathing zone was therefore larger than the ambient value. This was consistent with a number of experimental observations that the human thermal plume is capable of increasing the exposure risk by entraining particles from a lower level into the breathing zone (Rim and Novoselac, 2009; Bjørn and Nielsen, 2002).

Figure 5.13: Typical PM distributions yielded by the (a) E-E model and (b) the E-L model

(a) E-E model (Isosurface of \( CN = 0.1^* \))

(b) E-L model (Particle trajectories)

* CN is the normalized PM concentration in terms of the average PM concentration in the region of particle injection.
The PM concentration fields in the vertical plane \( Y = 2.25m \) predicted by the models are shown in Figure 14. Similar to the isothermal cases, the both models gave similar overall predictions. However, two distinct differences could be detected from Figure 5.14a and b. Firstly, thanks to the finer mesh size for model solution than that for post-processing of the Lagrangian particle trajectories, the E-E model generated a more smoothly distributed PM concentration field, which actually led to a higher resolution to present the local PM information. Although fine mesh could also be used for converting the Lagrangian particle trajectories, this would largely increase the computational cost. In addition, a finer mesh doesn’t necessarily mean a better solution of the PM concentration. It can be expected when the mesh for post-processing is as fine as that for model solution, the resultant PM concentration field would have an identical 3D distribution as the particle trajectories and thus lose the sense of concentration. On the contrary, a coarse mesh may not be able to provide sufficient numerical stability and resolution to present the PM concentration field. Based on Equation 5.17, the local PM concentration is related to the number of particle trajectories in a cell, however, this study revealed that when a cell containing a relatively large number of particle trajectories appears in the domain, an abnormally high PM concentration (e.g. \( C_N > 1000 \), not illustrated in the figures) would be generated in that cell. This poor numerical stability has made it quite challenging to find an appropriate mesh density and might be the reason for the small regions with high PM concentration near the ceiling, as observed in Figure 5.14b Zhang and Chen (2007b) and Zhao et al. (2008) proposed that for a given number of particle trajectories, there exists an optimal mesh structure for concentration calculation. Unfortunately, the criterion for judging an optimized mesh structure, especially for a complex geometry, is still absent due to the complexity depending on the number density of particle trajectories and the geometry of the domain. Further studies are still in demand to develop a more flexible algorithm to estimate the PM concentration based on the particle trajectories.

On the contrary, the E-E model is capable of giving a direct prediction to the PM concentration field, thus eliminates the uncertainties that might be caused by the post-processing procedures.
5.6 Conclusion

The Eulerian-Eulerian two-phase flow model was employed in this study to model the transport and concentration distribution of particulate matters. Computations were conducted with both transient and steady states, and both isothermal and thermal conditions. The model was validated using the experimental data available in the literature and compared against the existing two-phase flow models for PM transport, in the aspects of accuracy and computational cost. Conclusions arising from this study are as follows:

1. The Eulerian-Eulerian model has a comparable accuracy with the Lagrangian model and performs better than the drift-flux model, this is especially true when the particle concentration is relatively high and the particle size is large (e.g. PM10) when significant particle settling or deposition could happen.
2. When the PM concentration is preferred, the Eulerian-Eulerian model has its unique advantage over the Lagrangian model as it gives a direct prediction to the PM concentration, thus does not need any additional post-processing procedures. This not only largely reduces the computational cost, but also eliminates the uncertainties that might be caused by the additional post-processing procedures.
Chapter 6

Effects of Computational Thermal Manikin (CTM) Simplification on CFD Simulations

The main findings of this chapter have been published in:


6.1 CTM Simplifications Effects on Thermal Flow Field around Human Bodies

Abstract:
Simplified computational thermal manikins (CTMs) are beneficial to the computational efficiency of Computational Fluid Dynamics (CFD) simulations. However, the criterion of how to simplify a CTM is still absent. In this paper, three simplified CTMs (CTMs 2, 3 and 4) were rebuilt based on a detailed 3D-scanned manikin (CTM 1) using different simplification approaches. CFD computations of the human thermal plume in a quiescent indoor environment were conducted. The predicted airflow field using CTM 1 agreed well with the experimental observations from the literature. Although the simplified CTMs did not significantly affect the airflow predictions in the bulk regions, they strongly influenced the predicted airflow patterns near the CTMs. The predictive error of the CTM was strongly related to the simplification approach. The CTM generated from the surface-smoothing approach (CTM 2) was very close to CTM 1, while the required mesh elements for a stable numerical solution dropped by over 75%. Comparatively, the predictive errors of CTM 3 and 4 were considerable in the near-body regions. This study has illustrated the importance of keeping the key body features when simplifying a CTM. The surface-smoothing based simplification method was shown to be a promising approach.
6.1.1 Introduction

Micro-environment around human occupants in indoor spaces (Gao and Niu, 2004) has been attracting increasing research interest with concerns regarding personal thermal comfort and possible exposure to health hazards (Li et al., 2014b; Qian et al., 2008). The importance of the thermal buoyancy flows generated by human metabolic heat and its interaction with the environment has been thereby recognised. As the major thermal flows in most indoor spaces, the human thermal plumes can strongly affect the airflow pattern around human bodies (Craven and Settles, 2006). They play an important role in transporting the contaminants released from human bodies, such as pathogene-carrying droplets exhaled by coughing or sneezing (Chao et al., 2009; Zhu et al., 2006) and the gaseous and ultrafine particulate contaminants initiated from ozone reactions with human skin lipids (Wisthaler and Weschler, 2010; Rai et al., 2013). In addition, contaminants from near-floor levels could also be brought into the breathing zone by the thermal airflows and cause health issues (Aitken et al., 1999; Rim and Novoselac, 2009).

In order to evaluate the relationship between human bodies and their surroundings, computational thermal manikins (CTMs) representing human occupants have been included in the Computational Fluid Dynamics (CFD) models for indoor air quality, thermal comfort and exposure risk assessments. A number of CTMs varying from simple blocks and cylinders to detailed 3D-scanned manikins (Gao and Niu, 2004; Sorensen and Voigt, 2003; Martinho et al., 2012) have been reported in literature. Generally, detailed CTMs are desirable to use in CFD simulations in order to achieve better accurate predictions. Also, the inaccuracy in air velocity and temperature field predictions caused by model geometry differences could generate even greater errors in further CFD computations. However, the computational cost, on the other hand, increases exponentially when such complicated CTMs are used, which could become a barrier for practical applications. This is especially true when the interested space is occupied by multiple occupants, such as public transport vehicles (Rai and Chen, 2012) and classrooms (Wang et al., 2014b). As a compromise, CTMs have to be simplified in order to improve the computational efficiency, although this would inevitably reduce simulation accuracy.

As CFD approaches have been widely employed to assess the thermal comfort and estimate the health risks associated with contaminant exposure. Deevy and Gobean (2006) and Seo et al. (2013) investigated the effects of CTM simplification on CFD analyses of indoor airflow field and human thermal comfort. They reported that the predicted airflow field in the bulk region was not sensitive to the complexity of the
CTM, while the airflow field in the vicinity of the CTM was significantly changed when CTMs with different body complexity were used. The deviation in airflow field prediction further affected the results of thermal comfort evaluation. Therefore, a simplified CTM model seems to affect the airflow field prediction only in the regions close to the CTM. However, the inaccuracy caused by body geometry simplification could be greater when the CTMs are in motion. Mazumdar et al. (2011) simulated contaminants transport in an airliner cabin containing a moving passenger in the aisle. They used a rectangular block, a cylinder and a human-like block-set, respectively, to represent the moving passenger and found that the predicted concentration patterns were significantly different when different CTMs were employed.

Therefore, for the dual purposes of reducing the computational cost and maintaining an acceptable accuracy, appropriately simplified CTMs are generally preferred. Although a number of CTM simplification methods were reported in the literature, the criterion of choosing or creating an appropriately simplified CTM is still absent.

This study aimed to evaluate the effects of CTM simplifications on thermal flow field predictions by presenting detailed comparisons of various simplified CTMs. A smoothed 3D-scanned CTM, a skeleton based rebuilt CTM using Ruzic and Bi-kic (2014)’s method and a surface-area based CTM which was rebuilt according to Miyanaga et al. (2001)’s method were developed in this study. The numerical results yielded from those simplified models were tested and compared against the detailed 3D scanned model. The computational costs associated with the simplification methods were also analysed.

6.1.2 Methodology

6.1.2.1 Computational Domain and Boundary Conditions

The experimental data by Licina et al. (2014) was utilised in this study for model validation and comparison. During their experiment, a sitting manikin with a height of 1.23m, as illustrated in Figure 6.1a, was placed in the middle of an enclosed room with dimensions of 11.1m-length ×8m-width ×2.6m-height. The manikin was electrically heated and the ambient temperature was maintained at 26 °C. The ventilation system was turned off during the measurements to provide a quiescent condition. Thus the thermal buoyancy flow driven by the manikin body heat was the only airflow in the room. The airflow and temperature fields around the manikin surface were measured using the Particle Image Velocimetry (PIV) technique and complemented with the Pseudo Colour Visualisation (PCV) technique.
As the bulk air was free from the manikin’s thermal effect and was quiescent during the experiment, it is possible to apply a smaller computational domain during
the simulations. Model tests proved that stable predictions could be achieved when the domain dimensions reached $4m \times 3m \times 2.6m$. As shown in Figure 6.1b, the front, back and side walls of the computational domain were set as free-flow openings with zero gauge pressure to allow air flowing in and out depending on the interior conditions. The air temperature at the openings was set to be 26 °C according to the experimental setup.

6.1.2.2 The Computational Thermal Manikins (CTMs)

Four different CTM models including a 3D-scanned manikin, a smoothed 3D-scanned manikin, a skeleton-based CTM and a surface-area based CTM were employed in this study.

The 3D-scanned manikin model with detailed body and facial feature, designated CTM 1 in this study, was from an open database (http://www.ie.dtu.dk/manikin). This original manikin was slightly modified in order to achieve a comparable sitting posture and body surface area to the experimental manikin by Licina et al. (2014). As shown in Figure 6.1a, CTM 1 could overlap perfectly with the experimental manikin, except some local segments such as the hands and shanks. Thus the numerical results yielded from CTM 1 could be compared directly to the experimental data. In the following sections, CTM 1 was used as a baseline model for further developments of the other CTMs.

![Figure 6.2: Simplified and smoothed model (CTM 2) from the original 3D scanned model.](image)
The smoothed manikin (CTM 2) was obtained through smoothing CTM 1, as shown in Figure 6.2. Some unnecessary body and facial features including eyes, mouth and fingers were smoothed, while the overall and key features were preserved. In addition, errors from the 3D scanning process such as the abnormal shapes on the manikin hands were removed by smoothing the corresponding segments.

Ruzic and Bikic (2014) introduced a simple approach to build CTMs based on the skeleton structures of a human body. Their approach was employed in this study to build another thermal manikin model (CTM 3) based on the skeleton structure of CTM 1. To do this, the skeleton structure of CTM 1 was firstly extracted according to the major connections of each body segment, as well as the overall body shape (e.g. nose, elbows and knees). Some key sizes including the head dimensions, nostril size, shoulder width and leg diameter were measured. Then, the surface area of each body segment was measured. Finally, the body segments were rebuilt based on the skeleton structure and the segment surface area. The process of building CTM 3 was illustrated in Figure 6.3.

![Figure 6.3: Rebuilt manikin model (CTM 3) based on the skeleton structures.](image)

The surface-area based approach as introduced by Miyanaga et al. (2001) was employed in this study to build CTM 4. By using their approach, the 3D-scanned manikin (CTM 1) was firstly divided into several major body segments (head, upper body, left leg and right leg). Then the surface area and basic dimensions of each
segment were measured. Finally, the corresponding segment from CTM 1 was re-
built with similar surface area but using very simple geometries (i.e. cylinders and
rectangular solids). The schematic view of this approach is given in Figure 6.4.

![Figure 6.4: Highly simplified model (CTM 4) based on the key dimensions of the 3D
scanned model.](image)

The surface areas of all the aforementioned CTM models were listed in Table 6.1. In order to compare the body surface areas (BSAs) of the CTM models, all the
manikin models were divided into several body segments (head, body, arms and etc.)
and the corresponding weighting factor of each segment area was also listed in Table
6.1. The weighting factors of the same body segment area in all the CTM cases were
kept close to each other regardless of their differences in geometric features. The
overall BSA of the 3D scanned manikin (CTM 1) was $1.596m^2$, which agreed well
with Yu et al. (2010)'s anthropometric data that the female body surface area ranges
from $1.2m^2$ to $1.9m^2$ with a mean value of $1.522m^2$. The BSAs of the other three
CTM models were slightly changed after modification ($1.566m^2$ for CTM 2, $1.580m^2$
for CTM 3 and $1.638m^2$ for CTM 4). The overall and local percentage differences
of BSAs between each of the simplified models and the original CTM were listed in
Table 6.2. The biggest BSA difference occurred between CTM 1 and CTM 4 (less
than 3%), which was acceptable for engineering applications.
Table 6.1: Body surface areas and segment weighting factors of CTM models.

<table>
<thead>
<tr>
<th></th>
<th>CTM 1</th>
<th>CTM 2</th>
<th>CTM 3</th>
<th>CTM 4</th>
</tr>
</thead>
<tbody>
<tr>
<td>Area</td>
<td>Weighting factor</td>
<td>Area</td>
<td>Weighting factor</td>
<td>Area</td>
</tr>
<tr>
<td></td>
<td>(m²)</td>
<td>(%)</td>
<td>(m²)</td>
<td>(%)</td>
</tr>
<tr>
<td>HEAD</td>
<td>0.101</td>
<td>6.3</td>
<td>0.097</td>
<td>6.0</td>
</tr>
<tr>
<td>NECK</td>
<td>0.022</td>
<td>1.4</td>
<td>0.022</td>
<td>1.4</td>
</tr>
<tr>
<td>UPPER BODY</td>
<td>0.450</td>
<td>28.2</td>
<td>0.448</td>
<td>30.2</td>
</tr>
<tr>
<td>Arm L</td>
<td>0.159</td>
<td>10.0</td>
<td>0.151</td>
<td>9.8</td>
</tr>
<tr>
<td>Arm R</td>
<td>0.160</td>
<td>10.0</td>
<td>0.151</td>
<td>9.9</td>
</tr>
<tr>
<td>Leg L</td>
<td>0.352</td>
<td>22.1</td>
<td>0.349</td>
<td>20.9</td>
</tr>
<tr>
<td>Leg R</td>
<td>0.351</td>
<td>22.0</td>
<td>0.348</td>
<td>20.9</td>
</tr>
<tr>
<td>Total area</td>
<td>1.596</td>
<td></td>
<td>1.566</td>
<td></td>
</tr>
</tbody>
</table>

Table 6.2: Percentage differences of BSAs between the original model and the other models.

<table>
<thead>
<tr>
<th>Percentage Difference of Body Surface Area (%)</th>
<th>CTM 2 vs. CTM 1</th>
<th>CTM 3 vs. CTM 1</th>
<th>CTM 4 vs. CTM 1</th>
</tr>
</thead>
<tbody>
<tr>
<td>HEAD</td>
<td>-4.0</td>
<td>-5.3</td>
<td>-0.8</td>
</tr>
<tr>
<td>NECK</td>
<td>-0.5</td>
<td>-0.3</td>
<td></td>
</tr>
<tr>
<td>UPPER BODY</td>
<td>-0.5</td>
<td>6.9</td>
<td></td>
</tr>
<tr>
<td>Arm L</td>
<td>-4.7</td>
<td>-1.1</td>
<td>3.7</td>
</tr>
<tr>
<td>Arm R</td>
<td>-5.4</td>
<td>-1.5</td>
<td></td>
</tr>
<tr>
<td>Leg L</td>
<td>-0.8</td>
<td>-5.3</td>
<td>1.9</td>
</tr>
<tr>
<td>Leg R</td>
<td>-1.1</td>
<td>-5.2</td>
<td>2.1</td>
</tr>
<tr>
<td>Total area</td>
<td>-1.8</td>
<td>-1.0</td>
<td>2.7</td>
</tr>
</tbody>
</table>

6.1.2.3 Numerical Procedures

Since the thermal buoyancy flow in the domain was exclusively driven by the heat released from the CTM, proper specification of the heat flux at the CTM surface was crucial for predicting airflow field. Yan et al. (2009a) investigated the effects of the surface heat conditions of the CTM on the airflow field prediction. They found that both the total heating power of a CTM and the distribution of the heat flux on the CTM surface have significant effects on the predicted airflow pattern. They recommended that, in order to achieve a stable prediction when using simplified CTMs, the overall heating power of the manikin body should be kept constant. This
strategy was employed in this study in order to achieve a consistent heat condition with different CTM models.

During the experiment, a total heat load of 89W was applied at the manikin surface (Licina et al., 2014). However, heat transfer by radiation was not considered in this study. Thus, as recommended by Myrakami et al. (2000), only 40% of the heating power was applied on the manikin surface during the computations, which generated a total heating power of 35.6W for all the computational cases. Uniform heat fluxes were specified at the CTMs surface. The detailed heat load and heat flux at each body segment under various CTM models are showed in Table 6.3. Since the highly simplified model (CTM 4) used very simple geometry, the model divided the body into only four major segments (head, upper body and legs). The percentage differences of heat load for different body segments between the original and simplified CTMs are tabulated in Table 6.4. CTM 3 had the biggest difference of heat load at the upper body segment as compared to the original model (CTM 1), whilst CTM 2 had heat loads closest to those of CTM 1 at all body segments. The highly simplified model (CTM 4) also had heat loads similar to those of CTM 1 at the corresponding body segments.

Table 6.3: Head load and heat flux of CTM models.

<table>
<thead>
<tr>
<th></th>
<th>CTM 1</th>
<th>CTM 2</th>
<th>CTM 3</th>
<th>CTM 4</th>
</tr>
</thead>
<tbody>
<tr>
<td>HEAD</td>
<td>2.26</td>
<td>2.21</td>
<td>2.16</td>
<td>2.65</td>
</tr>
<tr>
<td></td>
<td>22.31</td>
<td>22.73</td>
<td>22.3</td>
<td>21.73</td>
</tr>
<tr>
<td>NECK</td>
<td>0.48</td>
<td>0.49</td>
<td>0.49</td>
<td>22.3</td>
</tr>
<tr>
<td>UPPER BODY</td>
<td>10.05</td>
<td>10.19</td>
<td>10.85</td>
<td>22.53</td>
</tr>
<tr>
<td>ARM L</td>
<td>3.54</td>
<td>3.44</td>
<td>3.54</td>
<td>17.34</td>
</tr>
<tr>
<td></td>
<td>22.31</td>
<td>22.3</td>
<td>22.3</td>
<td>21.73</td>
</tr>
<tr>
<td>ARM R</td>
<td>3.57</td>
<td>3.44</td>
<td>3.55</td>
<td>22.53</td>
</tr>
<tr>
<td>LEG L</td>
<td>7.85</td>
<td>7.93</td>
<td>7.51</td>
<td>7.80</td>
</tr>
<tr>
<td></td>
<td>22.3</td>
<td>22.73</td>
<td>22.3</td>
<td>21.73</td>
</tr>
<tr>
<td>LEG R</td>
<td>7.84</td>
<td>7.90</td>
<td>7.51</td>
<td>7.80</td>
</tr>
<tr>
<td>TOTAL heat</td>
<td>35.60</td>
<td>35.60</td>
<td>35.60</td>
<td>35.60</td>
</tr>
</tbody>
</table>

After a CTM was placed in the middle of the computational domain, the whole domain was discretised using unstructured tetrahedron mesh. Grid independence was tested by checking the mesh quality in ICEM (ANSYS, 2015) and the grid convergence index (GCI) (Roache, 1994; Ea and Hoekstra, 2014). The grid independency of the CTM 1 case was achieved at 2.5 million grid elements with the overall mesh quality
above 0.4 and the maximum y plus value less than 3 at the manikin surface. The same
process was applied to test the other three cases, while the mesh quality was kept
uniform to eliminate any possible errors induced by meshing. The total grid number
of three simplified cases varied from 1.9 million to 2.4 million. The grid number on
the surface of each CTM was given in Table 6.5.

Table 6.4: Percentage difference of heat load between the original model and the
other models.

<table>
<thead>
<tr>
<th></th>
<th>CTM 2 vs. CTM 1</th>
<th>CTM 3 vs. CTM 1</th>
<th>CTM 4 vs. CTM 1</th>
</tr>
</thead>
<tbody>
<tr>
<td>HEAD</td>
<td>-2.2</td>
<td>-4.3</td>
<td>-3.3</td>
</tr>
<tr>
<td>NECK</td>
<td>1.4</td>
<td>0.7</td>
<td></td>
</tr>
<tr>
<td>UPPER BODY</td>
<td>1.4</td>
<td>8.0</td>
<td></td>
</tr>
<tr>
<td>Arm L</td>
<td>-2.9</td>
<td>-0.1</td>
<td>1.1</td>
</tr>
<tr>
<td>Arm R</td>
<td>-3.6</td>
<td>-0.5</td>
<td></td>
</tr>
<tr>
<td>Leg L</td>
<td>1.1</td>
<td>-4.3</td>
<td>-0.6</td>
</tr>
<tr>
<td>Leg R</td>
<td>0.8</td>
<td>-4.2</td>
<td>-0.5</td>
</tr>
</tbody>
</table>

Table 6.5: Grid information for different CTM models.

<table>
<thead>
<tr>
<th></th>
<th>CTM 1</th>
<th>CTM 2</th>
<th>CTM 3</th>
<th>CTM 4</th>
</tr>
</thead>
<tbody>
<tr>
<td>HEAD</td>
<td>52204</td>
<td>12440</td>
<td>6970</td>
<td>990</td>
</tr>
<tr>
<td>NECK</td>
<td>3169</td>
<td>812</td>
<td>459</td>
<td></td>
</tr>
<tr>
<td>UPPER BODY</td>
<td>14832</td>
<td>3603</td>
<td>2184</td>
<td></td>
</tr>
<tr>
<td>Arm L</td>
<td>5150</td>
<td>1236</td>
<td>770</td>
<td>6442</td>
</tr>
<tr>
<td>Arm R</td>
<td>5147</td>
<td>1234</td>
<td>769</td>
<td></td>
</tr>
<tr>
<td>Leg L</td>
<td>10811</td>
<td>2674</td>
<td>1228</td>
<td>2542</td>
</tr>
<tr>
<td>Leg R</td>
<td>10795</td>
<td>2669</td>
<td>1226</td>
<td>2536</td>
</tr>
<tr>
<td>Tot. no. of grid elements</td>
<td>1.0E+05</td>
<td>2.5E+04</td>
<td>1.4E+04</td>
<td>1.3E+04</td>
</tr>
</tbody>
</table>
6.1.3 Results and Discussions

6.1.3.1 Model Validation

Numerical simulation was firstly preformed using the 3D-scanned CTM (CTM 1) to validate the CFD models. The predicted airflow field around the CTM (in the $Y = 0$ plane) is shown in Figure 6.5 and this was compared against the PIV image measured at ambient temperature of $26^\circ C$, given by Licina et al. (2014). As shown in Figure 6.5, the CFD simulation achieved a similar airflow development pattern to the experimental image in the vicinity of the manikin. As the thermal buoyancy flow develops upwards along the manikin skin, two significant thermal plumes were observed from both experimental measurements and the numerical results above the vertically postured body segments, the torso and the knees. Comparatively, the thermal flow above the near-horizontal thighs was much weaker. The local high-speed regions such as those in front of the chest and face were successfully captured. In Licina et al. (2014)’s experiments, higher air velocity was detected near the upper torso (Figure 6.5a) where the thermal plume detached itself from the manikin. Both the detachment and the high air velocity region were successfully predicted in the CFD simulations (Figure 6.5b). Although minor deviations between the PIV data and numerical results were detected in some bulk regions, the main effects of the thermal plume on the airflow acceleration near the manikin body were captured.

![PIV image by Licina et al. (2014) at 26°C](a)

![Velocity contour predicted using CFD at 26°C](b)

Figure 6.5: Comparison of overall velocity contour in front of the sitting manikin.
Licina et al. (2014) found in their experiments that the convective boundary layer of the thermal manikin fell within a range of $10 - 30\,mm$ from the surface of the manikin body. They measured the air velocities at 10 positions of different heights and $30\,mm$ offset from the manikin surface. The predicted air velocity at corresponding positions were extracted and compared against the experimental data, as shown in Figure 6.6, which demonstrated that the predicted air velocity at the selected positions agreed well with the experimental data. The average predictive error was approximately 3.9%, with the maximum value appeared at the lowest height location ($Z = 0.4m$). This local error was supposed to be induced by the slightly different crus angle between the experimental manikin and CTM 1.

Figure 6.6: Quantitative comparison of the air velocity at selected positions.

The predicted local airflow field in the breathing zone was also compared against the experimental data, as shown in Figure 6.7. Both the velocity magnitude and the vector directions were successfully predicted. According to Figure 6.7, the breathing zone was dominated by a significant uprising buoyancy flow. Higher air velocity up to $0.2m/s$ was found in the region between the chin and the nose tip, which agreed well with most experimental measurements reported in literature that the human thermal plume could induce a vertical air velocity of $0.1 - 0.2m/s$ in the breathing zone (Craven and Settles, 2006; Johnson et al., 1996; Rim and Novoselac, 2009).
The predicted airflow field agreed well with the PIV measurements given by Licina et al. (2014), both globally and locally. Thus, the CFD modelling of the thermal buoyancy flows driven by human body heat was proven effective and thereby a baseline for the following analyses and comparisons of CTM simplification could be established.

6.1.3.2 Effects of CTM Simplification

In order to assess the predictive errors associated with CTM simplification, further CFD simulations were conducted using the aforementioned simplified CTMs (i.e. CTMs 2, 3 and 4), with numerical procedures and boundary conditions exactly the same as those for CTM 1.

The predicted air velocity profiles along three lines penetrating the computational domain were firstly analysed, as shown in Figure 6.8. Lines 1 and 2 were horizontal lines above the manikin, while Line 3 was positioned to be inclined and parallel to the manikin torso so that it was partially located in the thermal plume. The results indicated that the simulated room could be basically divided into two distinct regions: the thermally-affected region and the bulk region. The thermally-affected region included the thermal convective boundary layer around the manikin and the thermal plume above it. The bulk region was free from the thermal effects of the CTM. According to Figure 6.8a and b, the simplification of CTM did not cause any noticeable change in the predicted airflow field in the bulk region (see the velocity profiles along Lines 1 and 2). However, the predicted air velocity profile in the thermally affected region was significantly changed when different CTMs were employed. This impact was especially significant in the thermal plume region above the manikin head. As
shown in Figure 6.8c, the predictive error of the local air velocity at m could be as large as 35% when CTM 3 was employed in place of CTM 1.

Figure 6.8: Velocity profiles along the selected lines.

Due to the inherent complexity of coupled numerical computations, CFD simulations of complicated indoor physical and chemical processes were generally conducted using a step-by-step approach. The air velocity and temperature fields were firstly solved to provide an optimal initial field, and then based on that, the transport of contaminants was solved using the Eulerian or Lagrangian approaches (Li et al., 2015b). The inaccuracy in the airflow field prediction could cause enlarged problems in the prediction of contaminant transport, which is especially true when the contaminants are released from the human body.

Figure 6.8 also reveals that all three simplified CTMs caused predictive errors relative to the baseline CTM. However, the magnitude of error varied with different CTMs. Comparatively, the prediction by CTM 2 was the closest to the baseline CTM.

The predicted air velocity and temperature fields in the vicinity of the CTMs are shown in Figure 6.9. For the convenience of comparison, the numerical results yielded from the baseline case (CTM 1) were also included. The visualised air velocity field (Figure 6.9a) and temperature field (Figure 6.9b) demonstrated that the thermal buoyancy flow developed upwards along the manikin surface. After departing from the manikin at the head, the flow kept accelerating until it reached its maximum velocity somewhere above the manikin head (Figure 6.9a). The comparison results indicate that the CTM geometry had a significant influence on the profile of the convective boundary layer. While CTMs 1, 2 and 3 yielded similar convective boundary layers
near the CTM surface, CTM 4 generated a notably different convective boundary layer. In the region immediately above CTM’s head and shoulders, CTM 4 predicted a much lower air velocity than those by the other three CTMs (Figure 6.9a) while the air temperature was largely over-predicted by CTM 4 in the same region (Figure 6.9b). Because simple blocks and cylinders were used to represent the CTM 4 manikin, inevitably some sharp corners and dead regions in the airflow would be resulted. Applying a block-based CTM is thought to be a major source of error for CTM 4.

(a) Air velocity field.

(b) Temperature field.

Figure 6.9: Effects of CTM simplification on prediction of velocity and temperature fields.

In addition, as the CTMs were slightly backward inclined, the buoyancy flow gained a horizontal velocity component. Thus, an inclined thermal plume was observed above the manikin head. Figure 6.9b shows that the thermal plume rose upwards, and then bended to the horizontal direction after hitting the room ceiling. When the overall developing processes of the thermal plumes were similar, the CTM geometry had a detectable effect on the thermal plume profile when different CTMs
were employed. As shown in Figure 6.9b, CTMs 1 and 2 generated almost the same thermal plume. The thermal plume yielded from CTM 3 had a larger horizontal velocity component, and consequently, it hit the room ceiling at a more downstream location. The thermal plume yielded from CTM 4 was significantly different from those of the other three cases, with the highest vertical velocity and the widest horizontal size of thermally affected region. Taking the numerical results generated with CTM 1 as the baseline, the predictive error induced by the simplified CTM was shown to increase with a degree of simplification.

The computations predicted that the maximum air velocity appeared at 20 to 26cm above the manikin head, which agreed well with most experimental observations (Craven and Settles, 2006; Sorensen and Voigt, 2003; Salmanzadeh et al., 2012) that the human thermal plume generally reaches its maximum speed 20 – 30cm above the occupant’s head in a low-turbulent room. By defining the thermal plume region as that in which the air velocity was greater than 0.2m/s, the outline profiles of the thermal plumes predicted using different CTMs at 25cm above the manikin head are plotted in Figure 6.10.

Figure 6.10: Thermal plume regions (velocity > 0.20m/s) at 25cm above manikin head.

Figure 6.10 reveals that the predicted shapes of the thermal plume regions were significantly different depending on the CTMs employed. At the selected plane, the thermal plume outline yielded from CTM 1 took a near-semicircular shape and was right above the manikin head. CTM 2 achieved a prediction very close to that of the baseline case due to its slight change of overall geometric features from CTM 1. On
On the other hand, CTMs 3 and 4 predicted distinctly different shapes and locations of thermal plume. The thermal plume predicted with CTM 3 was significantly deformed from the semicircular shape and was located closer to the manikin back. CTM 4 yielded a smaller elliptic shape which was closer to the front of the manikin than that predicted with CTM 1.

The areas of the thermal plume regions, as shown in Figure 6.10, were calculated and are shown in Figure 6.11. In terms of the overall thermal plume area at the selected horizontal plane, simulation with CTM 2 predicted the results closest to those of the CTM 1 case with an acceptable error of less than 6%. Also, by presenting the area of the thermal plume under various velocity ranges, over 55% of the illustrated thermal plume region was found in all cases with velocities ranging from $0.2m/s$ to $0.22m/s$. Under this main velocity range, the CTM 2 case also generated a very close prediction to that of the baseline case (CTM 1), with an acceptable error of less than 5% of the thermal plume area. In contrast, the errors of CTMs 3 and 4 were 14.8% and 18.1%, respectively.

Figure 6.11: Area of thermal plume region under various velocity ranges.

In summary, this study demonstrated that, although the simplified CTMs did not
notably affect the predicted airflow field in the bulk region, the characteristics of the predicted thermal buoyancy flow around the manikin were highly sensitive to the geometrical features of the CTM. Changing the CTM geometry would inevitably affect the outline of the thermal plume. Among the three simplified CTMs investigated in this study, CTM 2, which was obtained through smoothing the 3D-scanned manikin (CTM 1) and deleting unnecessary body features, achieved a satisfactorily accurate prediction of the thermal buoyancy flow with an error of less than 5%. Comparatively, the CTMs rebuilt based on the skeleton structure (CMT 3) and the surface area (CMT 4) caused significant predictive errors. Also, according to Table 6.5, the number of the mesh element on the CTM surface which was required to achieve a mesh-independent computation was reduced to over 75% after the surface smoothing (CTM 1 to CTM 2). This would be of great importance in reducing the computational cost of CFD simulations while retaining an acceptable predictive accuracy. For the rebuilt CTMs (CTM 3 and 4), considerable errors caused by simplification were the main drawbacks, although they could help further reduce the mesh density on the CTM surface. Therefore, the CTM simplifying approach through surface smoothing is more promising. This is especially true when the regions of interest are close to the occupants.

Simplifying CTMs through surface smoothing provides a robust approach to delete the redundant body features while preserving the key geometrical features, which would contribute to a largely improved computational efficiency without significantly sacrificing the quality of the numerical results. However, the smoothing approach presented in this study is not quantitative, and the degree of CTM simplification could be hard to control. Alternatively, the blooming computational geometry has brought out a number of advanced surface smoothing and simplifying algorithms such as the directed anisotropic diffusion algorithm (Banno and Ikeuchi, 2011) and the mesh decimating algorithm (Garland and Heckbert, 1997). These algorithms could simplify the local geometrical details by preserving the overall and key geometrical features. Therefore, there could be a quantitative and controllable approach to simplify the CTMs for CFD computations of the thermal buoyancy flows by human body heat, and these will be investigated in our future studies.

6.1.4 Conclusions

The thermal buoyancy flow driven by the human body heat in quiescent indoor air was simulated using a 3D-scanned CTM. The predicted airflow field agreed well with the PIV measurements by Licina et al. (2014). Comparisons of the predicted airflow
fields between the simplified CTMs and the baseline CTM have demonstrated that
the simplified CTMs did not have any detectable effects on the airflow prediction in
the bulk region. However, the predicted airflow in the thermally affected region was
highly sensitive to the approach of the CTM simplifications. The CTM generated
through surface smoothing (CTM 2) achieved a very close prediction as compared
to the baseline case with an error of less than 5%, whereas the predictive errors
associated with the skeleton-based (CTM 3) and the surface-area-based (CTM 4)
CTM simplifying approaches were 14.8% and 18.1%, respectively. Using the surface
smoothing approach, the required number of the mesh elements on the CTM sur-
face required to achieve mesh-independency could be reduced by 75%, which would
certainly contribute to an improved computational efficiency while maintaining a rea-
sonable predictive accuracy. Therefore, the CTM simplification approach based on
surface feature smooth was recommended for the future work.
6.2 Comparisons of Simplified CTMs on CFD Predictions under Different Ventilated Room

Abstract:
As one of the most basic parameters, manikin body feature could be an important factor influencing the airflow and temperature fields in indoor environments. This study aims to improve the computational efficiency by optimising and simplifying manikin body features. A 3D scanned computation thermal manikin (CTM) with extremely detailed body features was employed, followed by two simplified CTM models with different approaches. One of the simplified models was rebuilt based on the skeleton of the 3D scanned model with very limited body features, while the other model was simplified by removing some of the features from the 3D scanned model. All CTMs were tested under quiescent condition, followed by further comparisons under displacement and mixed ventilations. The outcomes indicated that the geometric difference of manikin body would have significant impact on the airflow patterns near manikin bodies, whilst it has very limited influence on the temperature field. The difference of body features could significantly affect the development of thermal plume, which mainly reflected above the manikin head. Also, change of CTM body features due to simplifications may become more sensitive to the predicted results under mixed ventilation, as a result of fewer interactions between the thermal plume and injected airflow.
6.2.1 Introduction

Indoor air quality and its potential impacts on the occupational health and safety are of increasing interests in recent years since most people spend over 85 percent of their time indoors (Lai et al., 2000; Salmanzadeh et al., 2012). The occupant bodies are expected to be the key factor affecting the thermal airflow patterns in built indoor spaces. In order to assist the investigations of occupant related thermal airflow characteristics in relation to the air quality, thermal comfort and potential indoor health risks, the inclusion of computer-simulated person (CTM) is essential in Computational Fluid Dynamics (CFD) simulations.

Most of the CTM models employed in the early of 2000s were extremely simple models that only contained the key outlines of human body (Myrakami et al., 2000; Hyun and Kleinstreuer, 2001; Hayashi et al., 2002). Extremely simple CTMs were widely employed to simulate multi-occupants indoor environments such as the hospital, classroom and public transports (Poussou et al., 2010; Qian et al., 2008), in order to minimise the computational cost. However, by using these CTMs, simulations could not be able to capture and predict detailed and accurate airflow and temperature characteristics at the interested regions are very close to the occupant bodies (e.g. breathing zone). As a temporary solution, the applied CTM models were lately improved with local refinement at corresponding body segments such as the head (Zhu et al., 2005; King Se et al., 2010). These local refined CTMs were mostly used to study the respiratory system related behaviours, which requires detailed facial features of occupants in order to provide detailed airflow information at the nostrils (Li et al., 2014b). Although this type of CTMs was beneficial to the local predictions, it was very time consuming to partially refine the CTM models and thereby was not widely applied. In recent years, a new type of CTM model that contains very detailed body features were increasingly used in some up to date researches (Nilsson et al., 2007; Martinho et al., 2012). The 3D scanned CPS provided full body features and very detailed geometry information, but it requires very fine grid to capture the surface features. The computational capacity would be the main barrier for further applications when the number of the 3D scanned CTMs is large and the computational domain is enlarged. In order to conduct simulations with a large number of CTMs, simplifications on the applied CTMs are necessary and a proper approach to simplify CTMs is strongly required.

Therefore, for the purpose of reducing computational cost without significantly sacrificing the numerical accuracy, this study aimed to assess the influence of CTM simplifications on the thermal airflow field using two simplified CTM models and to
provide recommendations for future applications. The 3D scanned CTM was initially employed to verify the reliability of CFD models. Then, two approaches were used to simplify and optimise the 3D scanned CTM model. The simulation results obtained from these developed CTM models were compared with the original CTM model in terms of airflow field and temperature profiles. The simplified CTM model with better accuracy and good computation efficiency would be recommended for future applications into multi-scale simulations that contains a large number of CTM models.

6.2.2 Numerical Methods

6.2.2.1 Computational Domain

The computational domain was based on the experimental setups and measurements with sitting manikin by Licina et al. (2014). The chamber environment was controlled to be quiescent during the experiment. In order to verify the numerical results, case 1 was set accordingly to Licina et al. (2014)’s experiment. The computational domain with dimensions of 4 m-length, 3 m-width and 2.6 m-height was tested to be sufficient to meet quiescent condition, as given in Figure 6.12. The front/back and side walls of the computational domain were set as openings with zero pressure to allow air flow in and out freely, while the air temperature inside the domain was the same as the experimental condition (26 °C). The surface area of the manikin model was controlled to be the same as the experimental manikin. The rate of total heat loss from the body was 89 W/m² in Licina et al. (2014)’s experiment. Since the rate of convective body heat loss governs around 40% of the total heat loss (Myrakami et al., 2000; Sorensen and Voigt, 2003), the convective heat flux of 35.6 W/m² was applied at the manikin skin while heat transfer by radiation was not included.

In order to further test the influence of the CTMs on the thermal airflow field under ventilated conditions, simulations were also conducted with two different ventilation schemes (i.e. the displacement (case 2) and mixed (case 3) ventilations). As illustrated in Figure 6.12, the inlet airflow with velocity of 0.15 m/s was released from a square plane (0.25 m²) behind the sitting CTM at near floor level and at the ceiling, for the displacement and mixed ventilations, respectively. The outlet was set at the top of the left side wall with zero pressure to allow air come in and out freely. The rest of the walls were set as solid walls rather than openings in case 2 and 3. All the tested CTMs were placed at the same location with the same sitting posture in all cases.
Figure 6.12: Computational domain with case 1: quiescent condition; case 2: displacement ventilation; and case 3: mixed ventilation.

6.2.2.2 Geometry of CTMs

The manikin model from open database (http://www.ie.dtu.dk/manikin) was employed in this study as the original model (OM). This model was a 3D scanned manikin model that has been widely used in other studies (Nilsson et al., 2007; Li et al., 2015c), due to its fully scanned and detailed body features. This CTM was modified in order to achieve the same leaning-back posture as the manikin model in Licina et al. (2014)’s experiment. The numerical outcomes from the original model have been validated in our previous study (Li et al., 2015c) through comparing with the experimental measurements by (Licina et al., 2014). Therefore, the simulation results from the OM were used as a reference to test other developed CTMs. The mesh decimating approach was initially utilised in our previous study to simplify the CTM model. Although the results proved the significance of body simplification on the thermal airflow predictions, the outcomes were limited to the mesh decimating approach only and cannot be used to assess those widely used simplification approaches in the literature. Therefore, in order to further investigate the effect of CTM simpli-
fication on the thermal airflow field in the indoor spaces, another two simplification approaches that have been widely reported in the literature (Myrakami et al., 2000; Poussou et al., 2010) were employed in this study to simplify the manikin models.

![Figure 6.13: The schematic views of the rebuilt manikin (RM) model (on the left), 3D scanned original manikin (OM) model (in the middle) and the simplified and smoothed manikin (SM) model (on the right).](image)

The first method (Ruzic and Bikic, 2014) was to completely rebuild the manikin model by following the same skeleton as the 3D scanned model (RM), as provided in Figure 6.13. The core skeleton structure from the 3D scanned model was extracted and applied as the reference to create the new model. The RM contains more regular surface curves and simpler body features, whilst very detailed body features were eliminated.

The second simplification approach was to maintain the overall body features of the 3D scanned model, but removing some insignificant features that have very limited impacts on the simulation results. It can be seen from Figure 6.13, the ears, eyes and mouth from the original 3D scanned model were eliminated from the simplified and smoothed model (SM). Also, redundant features such as the abnormal
segment on the fingertips were further removed by smoothing the manikin surface to reduce unnecessary grids.

6.2.2.3 Numerical Producers

The commercial CFD software CFX 14.5 (ANSYS, 2015) was employed to fulfill the simulations. The RNG k- turbulence model was applied in this study due to its high reputation on predicting three-dimensional airflow field in indoor environments (Chen, 1995; Chen et al., 2006). The discretisation for advection terms was based on the high order advection scheme to achieve better robustness and accuracy, while the SIMPLEC algorithm was applied to solve the pressure-velocity coupling. The scalable wall function was employed to resolve the boundary layer at near-wall regions. The computational domain and manikin surfaces were discretised using unstructured tetrahedral grids in the commercial software ICEM 14.5 (ANSYS, 2015).

Table 6.6: Grid information for different CTMs.

<table>
<thead>
<tr>
<th>No. of grid</th>
<th>OM</th>
<th>OM Coarse</th>
<th>RM</th>
<th>RM Coarse</th>
<th>SM</th>
<th>SM Coarse</th>
</tr>
</thead>
<tbody>
<tr>
<td>Head</td>
<td>52204</td>
<td>13210</td>
<td>6970</td>
<td>3669</td>
<td>12440</td>
<td>6424</td>
</tr>
<tr>
<td>Neck</td>
<td>3169</td>
<td>912</td>
<td>459</td>
<td>238</td>
<td>812</td>
<td>602</td>
</tr>
<tr>
<td>Upper body</td>
<td>14832</td>
<td>3776</td>
<td>2184</td>
<td>2968</td>
<td>3603</td>
<td>1879</td>
</tr>
<tr>
<td>Arms &amp; hands</td>
<td>10297</td>
<td>2503</td>
<td>1349</td>
<td>1539</td>
<td>2470</td>
<td>1247</td>
</tr>
<tr>
<td>Legs &amp; feet</td>
<td>21606</td>
<td>5296</td>
<td>2454</td>
<td>1161</td>
<td>5343</td>
<td>3879</td>
</tr>
<tr>
<td>Total no. of grid</td>
<td>1.0 ×</td>
<td>2.6 ×</td>
<td>1.3 ×</td>
<td>9.6 ×</td>
<td>2.5 ×</td>
<td>1.4 ×</td>
</tr>
<tr>
<td>at manikin surface</td>
<td>10^5</td>
<td>10^4</td>
<td>10^4</td>
<td>10^3</td>
<td>10^4</td>
<td>10^4</td>
</tr>
<tr>
<td>Total no. of grid</td>
<td>2.95 ×</td>
<td>2.65 ×</td>
<td>2.51 ×</td>
<td>2.45 ×</td>
<td>2.63 ×</td>
<td>2.54 ×</td>
</tr>
<tr>
<td>of the whole domain</td>
<td>10^6</td>
<td>10^6</td>
<td>10^6</td>
<td>10^6</td>
<td>10^6</td>
<td>10^6</td>
</tr>
</tbody>
</table>

Each CTM was computed under various combinations of grid sizes (e.g. OM with maximum surface elements of 5 mm and OM Coarse with maximum surface elements of 10 mm) to test and check the sensitivity of the grid size. The grid independence tests for all CTMs were conducted by checking the mesh quality in ICEM (ANSYS, 2015) and the grid convergence index (GCI) (Roache, 1994; Ea and Hoekstra, 2014). The overall mesh quality and GCI was controlled to be similar at the OM, RM and SM cases and the y+ values for all CTMs were controlled to be below 3. The tested grid number and size at bulk region were set to be uniform when testing different manikin models. Detailed grid information such as the number of grid for each studied case was in Table 6.6. The convergence with the residual less than 1 × 10^{-6} of the continuity equation was achieved within 1500 iterations.
6.2.3 Results and Discussion

6.2.3.1 Quiescent Condition (Case 1)

The numerical results regarding the global and local airflow field of the OM case was firstly presented and compared to the experimental measurements to validate the computational reliability.

Figure 6.14: Comparison of (a) the overall velocity contour in front of the sitting manikin; (b) the velocity contour and velocity vector at breathing zone between the experimental (left) and the simulation (right) results.

Globally, the overall velocity contour in front of the sitting manikin model obtained from the simulation was compared with the measurement by Licina et al in Figure 6.14a. Also, the comparison of velocity contour in conjunction with the velocity vector at the breathing zone of manikins was provided in Figure 6.14b. As can be seen in Figure 6.14a, the predicted velocity contour using OM in front of the sitting manikin body yielded very similar ascending airflow pattern to the experimental measurement. The results were slightly different in the regions far away from the manikin body. The experimental results seemed to be less consistent in some
bulk regions, probably due to the fact that the PIV images were not taken simultaneously during the experiment. For the airflow field in manikin breathing zone, the experimental measurements indicated very strong effect of buoyancy driven thermal plume generated by body heat that carries the air in the vicinity of manikin body travelling upward. This characteristic of airflow was also accurately predicted by the simulation, as shown in Figure 6.14b. Overall, the CFD approach (OM) predicted very similar results of the airflow field to the experimental measurements and the reliability of simulation was thereby validated. Further validations can be found in the published study using the same 3D scanned model and similar numerical setups (Li et al., 2015c).

Figure 6.15: Positions of selected lines with side view (X-Z plane) and top view (X-Y plane).

For the purpose of quantitative comparison, velocity profiles from various lines were extracted from each of the case and compared. As illustrated in Figure 4 (x-z plane view), three lines (L1, L2 and L3) were selected from the floor to the ceiling with same incline angle as the sitting manikin body. From the top view (x-y plane) in Figure 6.15, line 4 (L4) was selected to be half meter above the manikin head to capture the local of peak velocity, as well as the axially velocity distribution. The horizontal velocity profile just in front of the manikin nose was extract by line 5 (L5). Three additional cases (OM Coarse, RM Coarse and SM Coarse) were computed when comparing the velocity distributions along various locations. Each of these cases was computed with slightly coarsened finish. With the coarse model, much lower number of grids was required to compute the manikin surface, as listed Table 7.1. The mesh qualities for all the coarse models were still controlled to be uniform. The purpose of adding coarsened CTMs is to test the influence of CTM simplifications on the mesh sensitivity.
The results along L1 given in Figure 6.16 shows that from the ground level to the height of sitting manikin (0 - 1.2 m), the velocity profiles were very close among these six cases. At the region above the manikin head (above 1.2 m), the SM case was able to predict similar velocity profiles as the OM case, whereas the RM model predicted significantly different velocity distributions. The peak velocities at L1 predicted by the OM and SM model were 1.84 m/s and 1.78 m/s at height of 2.1 m, respectively. The difference of peak velocity between these two models was under 3%. On the other hand, the maximum velocity in RM model occurred at lower position (1.7 m) with magnitude of 0.14 m/s, which was dramatically different to the other models. L2 was showing a similar trend of velocity distribution as L1, while the difference between RM and the other cases was less significant. In the bulk region (L3), the velocity profiles were almost the same among various cases, although the velocities were still different at the ceiling level. Thus, the effect of manikin body regarding the airflow field would be significant in the vicinity of manikin body, but less obvious at bulk regions below manikin height. In terms of the bulk region above the manikin head, Figure 6.16 (L4) shows that the velocity difference occurred from about 1.5 m to 2.25 m, along x-axis, which is almost the region where the thermal plume effect is maximised. At the rest plots away from the thermal plume region, velocity distributions were predicted to be similar. Thus, the change of manikin body features would significantly affect the airflow field at the region above the manikin body where the thermal plume effect is strong. For the horizontal velocity distribution in front of the manikin nose (L5 in Figure 6.16), the predicted velocity profiles by different cases only vary in the vicinity of the manikin head with a diameter of 0.5 m. Since the thermal plume affected airflow was travelling mainly upward, the velocity difference along L5 could be mainly caused by the geometric difference of each model, which is insignificant from the observation of the plots.

When comparing each case with its coarsened model, it can be seen from Figure 6.16 (L1 and L2) that the velocity predictions were very stable between the RM and RM Coarse cases, although the RM model did not agree well with other models. This indicated that the skeleton based simplifications had less impact on the mesh sensitivity. On the contrary, for the SM model which obtained closer results to the OM case, the numerical outcomes would be very sensitive to the simplification level and mesh quality.
Also, the temperature contours of three studied cases were compared in Figure 6.17. All cases predicted very obvious thermal plume above the manikin heads, while less obvious thermal plume development can be noticed above the manikin knees. However, the developing pattern of thermal plume from the RM case was slightly different to the other two cases. The backward development of thermal plume pre-
dicted by the RM case was more significant. This could be caused by the change of surface features and projection curves of the RM case. The secondary thermal plume generated by the heat of lower manikin body travelled upward with similar pattern as the major one in same case but less intense.

Figure 6.17: Temperature contour for studied cases.

6.2.3.2 Displacement and Mixed Ventilations (Case 2 and 3)

The influence and significance of the CTM simplification on the airflow field may vary when the ventilation schemes changes. The simplified CTMs (RM and SM) were further compared to the original model under the displacement and mixed ventilations that are the most commonly used ventilation schemes in indoor spaces such as office, classroom and etc. Since the aforementioned results indicated that the CTM simplification had higher impact on the airflow field than the temperature profiles, focused were drawn mainly on the velocity field after the HVAC system was considered.

The velocity vectors representing the overall airflow patterns predicted by all the CTM cases under different ventilations were compared at the mid-plane (X-Z plane), as shown in Figure 6.18. Under the displacement ventilation system, the inlet airflow travelled horizontally across the near-floor region until it reached the thermally affected region in the vicinity of the sitting CTM. By interacting with buoyancy driven thermal plume, injected airflow at relatively higher level was interrupted and changed its direction from horizontal to nearly vertical. Obvious ascending pattern of the airflow can be observed in front of and above the CTM body. The air exchange rate is relatively higher at the occupant’s breathing zone under displacement ventilation thanks to the interactions between the injected airflow and the thermal plume, although this may potentially brought near-floor level contaminants into the breathing zone as well. On the other hand, when the ventilation was switched to the mixed scheme, the airflow field completely changed. Since the inlet velocity was
not significantly high, after reaching the floor, the injected airflow quickly dispersed and stayed at the near-floor level, which suppressed it interactions with the thermal plume generated by the heated CTM. As a result, the airflow was divided into two main streams by the injected airflow and thermal plume, respectively, which was not ideal for even air distribution and exchange.

![Figure 6.18: Velocity vectors predicted by different CTMs under the displacement (left) and mixed (right) ventilations.](image)

By comparing the predicted airflow field using various CTMs, the impact of the body simplifications on the airflow field can be clearly visualised under both ventilation schemes, as demonstrated in Figure 6.18. With the displacement ventilation, it seemed that the change of body features had less impact on the over airflow pattern than that with mixed ventilation. This is probably because that the airflow velocity around the sitting CTM was relatively high and thereby the effect of the thermal

116
plume became less significant. In terms of the mixed ventilation, the predicted velocity around the CTM by the SM case agreed better to the OM than that of the RM case. Both OM and SM cases obtained relatively wider thermally affected region around the CTMs than the rebuilt model.

Figure 6.19: Velocity profiles at selected lines under displacement ventilation.
Figure 6.20: Temperature profiles at selected lines under displacement ventilation.
The quantitative velocity profiles at the same selected lines as aforementioned (Figure 6.15) were compared among all the CTMs under studied ventilations. Under displacement ventilation, it can be noticed from Figure 6.19 that the plotted velocity profiles by the SM case were very close to the original model, despite some minor differences at L2. The skeleton based rebuilt CTM case, on the other hand, failed to predict similar local velocity distributions to the OM case particularly at L2 and L3, although it agreed well with the original model on the global velocity distributions. It seemed that the impact of body simplification was quite significant at L2, which was placed 25 cm in front of the CTM upper torso, while it also had considerable influence on the other two lines (L1 and L3) inside the thermally affect regions. After changing the ventilation to mixed scheme, the deviations of velocity profiles (Figure 6.21) at L2 from both SM and RM cases were enlarged, although the velocity patterns were remained similar among all studied CTMs. Despite that, under mixed ventilation scheme, the RM case also failed to capture the similar velocity profiles along the longitudinal direction (L4), in which the deviation still occurred at the thermally affected region very close to the CTM body. Therefore, according to the comparison, it can be concluded that the SM model was more promising to predict similar airflow profiles globally and locally to the original model than the rebuilt model, while the influence of CTM simplifications was more obvious under the mixed ventilation scheme.

In terms of the temperature distribution, it can be noticed from Figure 6.22 and 6.22 that the main deviations caused by CTM simplifications occurred at Line 1 and 2 under both ventilation schemes, whereas the temperature profiles were not significantly affected by different CTMs along the longitudinal direction (Line 4) and the horizontal direction (Line 5). The surface-smoothed CTM (SM) managed to obtained very close temperature distributions to the original models at all selected lines, although the temperature magnitudes were slightly different (with error less than 3%) at Line 1 and 2. Under mixed ventilation scheme, the rebuilt model (RM) did not predict very similar temperature distribution to the OM in the vicinity to the CTM (Line 1), which could be caused by the body feature differences at local body segments. Generally, the effects of the CTM simplifications on the temperature fields were less significant than that under the airflow fields.
Figure 6.21: Velocity profiles at selected lines under mixed ventilation.
Figure 6.22: Temperature profiles at selected lines under mixed ventilation.
6.2.4 Conclusions

The effect of CTM diversity by simplifications on predicting the thermal airflow fields were studied under quiescent condition, displacement and mixed ventilations. Based on the outcomes, the conclusions rising from this study are as follows:

The geometry of CTM has more significant effect on the local airflow field around the manikin body than that on the temperature distributions. The significance of manikin model variety will be enlarged on top of the manikin head due to the effect of buoyancy driven thermal plume by body heat, while the locations of the maximum velocity were very sensitive to the applied CTMs.

The SM model is more capable of obtaining reliable and accurate predictions to the original model with reduced computational cost. Thus, the SM model is recommended to replace the original model when studies require very detailed body features. However, the grid independence of this simplification approach is quite sensitive. The skeleton based model (RM), however, was not as good as the SM on predicting the local airflow field, but it required much less computation resource and had better numerical stability, It is preferred for simulations with high amount of CTMs.

The geometrical diversity of CTM caused by simplifications had higher impact on the thermal airflow field under the mixed ventilation than that with displacement ventilation based on the studied cases. This outcome may vary if the vent sizes and positions are changed or the inlet air velocity is different. Since the focus of this study is on the CTM simplification approaches, further test of HVAC systems in conjunction with CTMs will be conducted in the future studies.
Chapter 7

Development and Evaluations of Simplified Manikins in Cabin Environments

The main findings of this chapter have been published in:


7.1 A Novel and Quantifiable Manikin Simplification Approach for Occupied Cabin Environments

Abstract:

This study presented an iterative approach to simplify computational thermal manikins (CTMs) based on the mesh decimating algorithm (Garland and Heckbert, 1997). The approach could largely simplify 3D-scanned manikins while maintaining their key geometrical features. The level of simplification could be quantified through controlling the iteration number of simplification. CFD computations of human thermal plume in a quiescent room were performed using CTMs with different levels of simplification. The numerical results were compared against the experimental data available in the literature. The results demonstrated that within the scope of this study, the CTM simplification only affected the predicted airflow field in the thermally-affected regions where the normalized air velocity was larger than 0.5. The predictive error increased with the dimensionless simplification index (SI). When SI was less than $3.5 \times 10^{-4}$, the error induced by CTM simplification could be safely ignored. Contaminant transport in a densely occupied airliner cabin section was also simulated using the simplified CTMs. The results revealed that although the CTM simplification only affected airflow field prediction of the thermal plume regions, it could enlarge the predictive error of contaminant transport in the whole computational domain. In addition, this study found that the CTMs yielded from the algorithm were more numerically stable in terms of CFD computations.
7.1.1 Introduction

As a cost-efficient approach, computational fluid dynamics (CFD) has been widely employed in the research area of ventilation and air quality in indoor environments. As an important component, computational thermal manikins (CTMs) representing human occupants have been included in many CFD models. In a realistic indoor environment, the human occupants interact closely with their surroundings by serving as obstacles of airflow, source and sink of various contaminants, and the major heat source of thermal buoyancy flows (Zukowska et al., 2012). In order to achieve an effective CFD prediction associated with ventilation performance, air quality, thermal comfort or exposure risk assessment, an appropriate characterization of the human occupants in a CFD model is very important.

During the past years, numerous CTMs have been employed varying from simple blocks and cylinders to 3D-scanned manikins. The CTMs available in the literature could be basically classified into 3 categories:

1. Simple CTM models (Craven and Settles, 2006; Yan et al., 2009a; Mazumdar et al., 2011; Rai and Chen, 2012; Villi and De Carli, 2014), which use simple geometries (e.g., cylinders, spheres and rectangular blocks, etc.) and their combinations to represent human bodies, are the simplest approximation of the human occupants.

2. Human-like CAD models (Kilic and Sevilgen, 2008; Ztek et al., 2010; Zhang et al., 2012; Seo et al., 2013; Ruzic and Bikic, 2014), which are built based on the human skeleton structures (Ruzic and Bikic, 2014) using CAD codes, have identifiable head, torso, arms and legs. However, detailed body features such as eyes, nose, fingers and toes are generally ignored.

3. 3D-scanned manikins (Sorensen and Voigt, 2003; Gao and Niu, 2004; Martinho et al., 2012), which are reconstructed from 3D scans of full-size dummies, have detailed body and facial features and are the most accurate representation of human occupants.

The simplified CTMs could significantly reduce the computational cost while the detailed CTMs are able to contribute to an improved accuracy of prediction. However, the question how to select a compromised CTM based on the specific requirements still remains unanswered, despite this issue has been widely recognized and some efforts (Yan et al., 2009a; Deevy and Gobeau, 2006) have been devoted to seek the
answer. As a quantitative guideline to an appropriately simplified CTM is absent, the CTMs available in the literature differed a lot from each other and the development of these CTMs was quite arbitrary.

Using three CTMs with different resolution levels of body features, Deevy and Gobeau (2006) analyzed the effects of CTM geometry on CFD simulations of airflow field in a ventilated room. They reported that for the bulk region, the simplified CTMs (Category 1 and 2) returned very similar air velocity fields to that yielded from the CTM with detailed body features (Category 3). However, significant differences were found in the regions close to the CTM surface. This was consistent with the conclusion drawn by Seo et al. (2013), who also found that more precise results were obtained for the evaluation of thermal comfort when a detailed CTM was used. Therefore, it could be expected that simplified CTMs may be sufficient for predictions of the global flow field, while detailed CTMs would be preferred when the near-occupant regions or the occupants themselves are concerned.

It should be noted that the scenarios of Deevy and Gobeau (2006) and Seo et al. (2013) were quite simple, i.e. unfurnished rooms containing a single occupant in the middle and excluded the interactions between multiple or moving occupants. In recent years, CFD has been widely utilized in relevant studies with the increasing concerns on the health risks associated with communicable diseases (e.g. SARS and flu, etc.) in public transport aircraft/vehicle cabins (Olsen et al., 2003; Furuya, 2007). In a densely occupied narrow indoor space such as an airliner cabin, the bulk region free from the occupant effects could be very small, thus the human thermal plumes could overlap and the predicted results would be highly sensitive to the CTM geometry. Rai and Chen (2012) simulated ozone distribution in an airliner cabin section using two different CTMs and found that the predictive error of ozone concentration in the passenger breathing zone could be as large as 15%. Mazumdar et al. (2011) investigated the effects of passenger movement on contaminant transport in an airliner cabin. A rectangular block, a cylinder and a human-like block-set were used in their study to represent the moving passenger, respectively. Significantly different patterns of contaminant distribution were predicted for each.

Furthermore, due to the strong non-linear characteristics of contaminant/pathogen transport in aircraft/train cabins (Olsen et al., 2003), a full-cabin CFD model containing dozens or even hundreds of CTMs is often necessary in order to achieve an all-sided prediction. However, this would largely increase the computational cost. For such large-scale computations, it is not practical to use 3D-scanned manikins
when one considers the computational efficiency. Therefore, choosing appropriately simplified CTMs is crucial for optimizing the efficiency and accuracy.

In recent years, a number of geometry-simplifying algorithms, such as the mesh decimating algorithm (Garland and Heckbert, 1997) and the polygonal surface simplification algorithms (Hagbi and El-Sana, 2010), have been proposed and widely utilized in the fields of computational geometry. These algorithms have contributed to a significantly reduced computational time of geometry processing through deleting unnecessary geometrical complexities (Daneshpajouh et al., 2012). Applying these algorithms to simplify CTMs in the CFD models of occupied indoor spaces will certainly help reducing the computational cost without significantly sacrificing the quality of results. Therefore in this study, one of the highly reputed feature-preserving simplification algorithms - the mesh decimating algorithm by Garland and Heckbert (1997) - was employed to simplify laser-scanned manikins. Several CTMs with different levels of simplification were obtained and then incorporated into the CFD models of sparkly and densely occupied indoor spaces. The thermal flow fields and contaminant concentration fields were predicted and the error associated with CTM simplification was analyzed. A quantitative criterion was recommended for choosing an appropriate CTM model.

7.1.2 Methods

7.1.2.1 The Mesh Decimating Algorithm

Garland and Heckbert (1997) proposed a mesh decimating algorithm based on quadric error metrics, which could significantly simplify a complex geometry while preserving the primary features of the object. According to Garland and Heckbert (1997), a 3D geometry surface could be represented by a number of triangular faces. The basic idea of mesh decimation is to contract the pairs of triangle vertices, as illustrated in Figure 7.1. The operation of pair contraction \((v_1, v_2) \rightarrow v\) moves the vertices \(v_1\) and \(v_2\) to a new position \(v\), rebuilds the triangles by connecting all their incident edges to \(v\) and then deletes \(v_1\) and \(v_2\).

The key job of the mesh decimating algorithm is to determine an optimal position of \(v\). As shown in Figure 7.1, the new vertex \(v(x, y, z)\) and the original triangular planes \((p_i(ax + by + cz + d = 0))\) could be expressed in the form of matrices by, respectively.
Figure 7.1: The mesh decimating algorithm by Garland and Heckbert (1997).

\[ v = (x, y, z)^T \]  
(7.1)

and

\[ p_i = (a, b, c, d)^T \]  
(7.2)

Thus, the sum of squared distances of \( v \) to the planes \( (i = 1 - N) \) is:

\[ \Delta (v) = v^T \left( \sum_{i=1-N} K_{pi} \right) v \]  
(7.3)

where, \( K_{pi} \) is the matrix:

\[
K_p = p_ip_i^T = \begin{bmatrix}
  a^2 & ab & ac & ad \\
  ab & b^2 & bc & bd \\
  ac & bc & c^2 & cd \\
  ad & bd & cd & d^2
\end{bmatrix}
\]  
(7.4)

This fundamental error quadric \( K_p \) can be used to find the squared distance of any point in space to the plane \( p_i \). For the planes \( (i = 1 - N) \) as shown in Figure 7.1, the sum of their fundamental error quadrics makes a new single matrix \( Q \). Therefore, equation 7.3 is rewritten by

\[ \Delta (v) = v^T Qv \]  
(7.5)

In order to minimize the error, the position of \( v \) should satisfy:
The aforementioned mesh decimating algorithm was applied to simplify laser-scanned manikins in this study. The original manikin model with detailed body features was downloaded from the open database http://www.cfd-benchmark.com. The simplification was iteratively performed, with the following criteria of judging a valid vertex pair \((v_1, v_2)\) for contraction (Garland and Heckbert, 1997):

1. \((v_1, v_2)\) is an edge of a triangle (Figure 7.1a), or
2. \(\|v_1 - v_2\| < t\), where \(t\) is a threshold parameter for non-edge pair contraction (Figure 7.1b).

To begin with, the manikin model was divided into 250,000 initial triangular faces, which were sufficiently fine to fulfill an accurate capture of the dummy geometry. Then, a target percentage of reduction \((\Phi = 0.8)\) was set for each iteration in order to achieve a smooth simplification. Thus, the CTM simplification could be quantified using a dimensionless simplification index:

\[
SI = \frac{1}{(N_0 \varphi^n)}
\]

where, \(N_0\) is the initial number of the triangular faces and \(n\) is the iteration number of CTM simplification. \(SI\) indicates the ratio of the mean area of a single triangular face to the total CTM surface area. Obviously, an elevated \(SI\) means larger triangular faces. Through controlling \(SI\) or the iteration number, the mesh decimating algorithm provides a quantitative and controllable approach to simplify CTMs.

The simplified CTMs generated from the algorithm are shown in Figure 7.2. With increasing iteration number, the body and facial details were gradually deleted as the small triangular faces were merged into larger ones. As shown in Figure 7.2, the CTM geometry was only negligibly changed after 10 iterations. When the iteration reached 20, the CTM head still had identifiable face features. However, as the iteration number increased up to 30, the CTM head was simplified into a simple block without identifiable facial features. In order to avoid excessive simplifications, the CTMs
generated from over 30 iterations were excluded from this study as their body features were overly deleted.

Figure 7.2: Simplification of the laser-scanned CTM model through the mesh decimating algorithm with different numbers of iteration.

Table 7.1: Quantification of the CTM simplification.

<table>
<thead>
<tr>
<th>Number of iterations</th>
<th>0</th>
<th>10</th>
<th>20</th>
<th>25</th>
<th>30</th>
</tr>
</thead>
<tbody>
<tr>
<td>Simplification index</td>
<td>4.0E-6</td>
<td>3.7E-5</td>
<td>3.5E-4</td>
<td>1.1E-3</td>
<td>3.2E-3</td>
</tr>
<tr>
<td>Number of triangular faces</td>
<td>250,000</td>
<td>26,844</td>
<td>2882</td>
<td>944</td>
<td>309</td>
</tr>
<tr>
<td>Surface area (m²)</td>
<td>1.581</td>
<td>1.579</td>
<td>1.577</td>
<td>1.568</td>
<td>1.530</td>
</tr>
</tbody>
</table>

All the 30 iterations were finished in only 2 minutes on an ordinary desktop computer, which indicated the high efficiency of the algorithm. Table 7.1 shows the parameters relevant to the CTM simplification. The number of triangular surfaces composing the CTM decreased down to around 10% of its original value within only 10 iterations. When the iteration number reached 30, the triangle number decreased by 3 orders of magnitudes (250,000 down to 309). On the other hand, the surface area of the CTM only decreased slightly from 1.581m² to 1.530m² over the 30 iterations, with an acceptable maximum error of 3.2%. The CTM surface area is one of the most
important parameters affecting CFD predictions of the human micro-environments. Thus, the CTMs generated from the algorithm of this study could minimize the uncertainties induced by the changed surface area. In addition, as shown in Figure 7.2, the general shape of the CTM was still preserved even the iteration number was as many as 30. Therefore, compared with the Category 1 and 2 CTMs, the CTMs generated from the algorithm are more geometrically constant with realistic occupants.

7.1.2.2 The CFD Computations

The experimental data by Licina et al. (2014) were selected for model validation and comparison. In their experiments, a seated thermal manikin was placed in the middle of a test room. The manikin with a constant skin temperature of 32°C was the only heat source driving a thermal buoyancy flow in the room. The airflow field near the manikin surface was measured using the particle image velocimetry (PIV) technique. In order to eliminate the edge-effects of the room walls on the airflow field in the vicinity of the manikin, a large test room (11.1m-width 8.0m-depth 2.6m-height) was constructed. In order to control the computational cost, a smaller computational domain was built and its side walls were set as openings with a constant temperature of 26°C. This eliminated edge-effects and allowed mass flowing in or out of the domain based on the flow conditions in the domain. The computations proved that when as domain size reached 3.0m-width 4.0m-depth 2.6m-height (Figure 7.3), the computational results became stable.

Figure 7.3: The computational domain based on Licina et al. (2014).
In order to achieve a comparable sitting posture and a close surface area to those in the experiments (Licina et al., 2014), the original CTM model was slightly modified so that a good overall overlapping between the experimental manikin and the CTM was achieved. Separate computations were conducted using different CTMs as illustrated in Figure 7.3, respectively, with exactly the same boundary conditions and numerical procedures.

Based on the above conditions, the steady-state, incompressible Navier-Stokes equations (ANSYS, 2015) were selected for the airflow field, together with the Boussinesq approximation (ANSYS, 2015) for the thermal buoyancy flows. Transport of gaseous contaminant around the CTMs was also simulated in this study (see Section 7.1.3.2), using the transport equation of a transportable scalar (ANSYS, 2015)

\[
\frac{\partial}{\partial t} (\rho C) + \nabla \cdot (\rho \mathbf{U} - D \nabla C) = S_C
\]  
(7.8)

where \(C\) is a scalar representing the contaminant concentration, \(D\) is the kinematic diffusivity of the contaminant in air and \(S_C\) is the source term.

The model equations were solved using the commercial CFD code ANSYS-CFX 14.5 (ANSYS, 2015), with the RNG k-\(\varepsilon\) model for the air turbulence because of its successful applications in simulating the thermal environments around human bodies (Gao and Niu, 2005).

In order to solve the model equations, the computational domains were discretized using unstructured mesh. Mesh independence was achieved for all the models using the same CTM and volume mesh sizes. The original computational mesh was taken as the standard mesh (1× standard mesh) and then coarsened by a factor of 2× and 4×, respectively. The mesh sensitivity and numerical stability of a simplified CTM (20 iterations) were whereby analysed and compared to those of the original CTM. Figure 7.4 shows the predicted air velocity profiles along a vertical line (Line 1) above the CTM head. The results revealed that significant deviation was caused when the mesh of the original CTM was coarsened by a factor of 2× (Figure 7.4a). The computation failed to converge when the mesh was further coarsened by a factor of 4×. On the contrary, when the simplified CTM was used, a coarsening factor of 2× did not cause any measurable error in the predicted velocity profile (Figure 7.4b). The error was still acceptable even when the coarsening factor increased up to 4×. It was supposed that the merged triangular faces largely eliminated the unnecessary complexities generated from 3D-scans, which helped improving the mesh quality and the numerical stability. The coarsened CTM mesh further reduced the number of
mesh elements and hence the computational cost, which allowed more cost-efficient CFD simulations, but without much sacrifice of the accuracy.

Figure 7.4: The effects of mesh coarsening on different CTMs (Line 1).

7.1.3 Results and Discussion

7.1.3.1 Model Validation and Analysis

The predicted airflow field in the Y = 0 m plane is shown in Figure 7.5a. The computations revealed that a significant thermal buoyancy flow was induced by the occupant body heat. The top velocity above the CTM head was approximately 0.2 m/s, which was consistent with previous experimental measurements in the literature that the human thermal plume can produce vertical air velocities up to 0.25 m/s (Craven and Settles, 2006; Johnson et al., 1996; Homma and Yakiyama, 1988; Rim and Novoselac, 2009). The predicted air velocities were measured at different heights in the vicinity of the manikin (3 cm offset from the manikin surface, as illustrated by the red dots in Figure 7.5a and compared in Figure 7.5b against the experimental data by Licina et al. (2014). The results showed that the predicted air velocities agreed well with the experimental data, indicating the effectiveness of numerical procedures of this study. Figure 7.5b also indicates that the predicted airflow field could be different depending on the iteration number of CTM simplification. It seemed that for the case of this study, the iteration number of 20 was a threshold, as no significant difference was detected between the predictions when the number of iteration did not exceed 20. However, deviation appeared and was then enlarged beyond the threshold.
Figure 7.5: Numerical prediction of the overall airflow field.

The predicted airflow field in the breathing zone was also compared against the PIV data of Licina et al. (2014), as shown in Figure 7.6. The simulations achieved a very close airflow field to that obtained experimentally. Figure 6 shows that the breathing zone was dominated by a significant upwelling airflow, with higher velocity immediately next to the manikin mouth and nose. In order to achieve a quantitative comparison, the air velocity profile along a horizontal line (Line 6) in the breathing zone, which started from the nose and had a length of 150 mm (Figure 7.6a and 7.6b), was extracted from the PIV image using a Matlab code and compared against the numerical results (Figure 7.6c). Similar to Figure 7.5, the predicted air velocity field was very stable and agreed well with the experimental data when the iteration number of CTM simplification did not exceed 20.
Further analyses were performed in this study to quantify the affecting range of CTM simplification. Figure 7.7 shows the air velocity profiles along 5 lines (Line 1 - 5, Figure 4) originating from the midpoint of an imaginary line joining the ears. The midpoint is defined as the central point of a personal breathing zone in the Australia Worksafe standard NOHSC 3008 (Australia, 2013). According to Figure 7.7, the CTM simplification within the scope of this study did not cause any measurable change in the predicted airflow field in the bulk regions, regardless of the iteration number. Figure 7.7b -7.7e show that the maximum affecting range in the horizontal directions was approximately 0.6 m from the breathing zone centre. Beyond this distance, the airflow prediction was free from the effects of CTM simplification. How-
ever, significant error was detected in the vicinity of the manikin when the iteration number exceeded 20. As the radius of a personal breathing zone is generally taken as 0.3 m (Australia, 2013), it could be expected that the whole breathing zone was in the affected range of CTM simplification. Therefore, significant error may occur when one simulates inhalation or contaminant exposure using an inappropriately simplified CTM. Comparatively, this study presented a quantitative and controllable CTM simplification approach to minimize this error.

Figure 7.7: Airflow in the heat-affected zone is sensitive to the CTM simplification.

Figure 7.7a differed from Figure 7.7b - 7.7e and demonstrated that the CTM simplification had a larger affecting distance in the vertical direction (Line 1, over 1.5 m from the breathing zone centre). A comprehensive analysis of Figure 7.7 suggested that the effects of CTM simplification on airflow field prediction were mainly detected in the region of thermal plume, as the thermal plume existed mainly in the regions above the occupant in quiescent air (see Figure 7.7a).
In order to quantify the affecting range of CTM simplification, the air velocity profiles yielded from the original CTM were taken as the baselines and the error was analyzed in terms of the iteration number and SI, as shown in Figure 7.8. Figure 7.8 shows the relative errors in terms of the local air velocity, where the air velocity was normalized based on the maximum air velocity in the thermal plume. When the local air velocity was less than half of the maximum velocity, the predictive error was negligible. However, the error increased sharply as the normalized velocity exceeded 0.5 and could be larger than 15% in some local areas as the iteration number increased up to 30. Figure 7.8 demonstrates that significant error existed along Line 1 and Line 2 (represented by the pink and black symbols), which were above and in front of the CTM, respectively. When the regions in which the dimensionless normalized velocity exceeded 0.5 were termed as the thermally affected regions (TARs), the TARs are the critical areas deserving special considerations when investigating indoor human thermal environments using simplified CTMs.

Figure 7.8: Predictive error caused by CTM simplification.
7.1.3.2 Application of Simplified CTMs in an Airliner Cabin Section

The effects of CTM simplification on the predictions of contaminant transport in a densely occupied indoor space (e.g., an airliner cabin section) were also studied. By assuming a symmetric distribution of the airflow field across the aisle plane and a periodic distribution along the axial direction, a typical medium-size airliner cabin could be represented by a cabin section containing 3 seat and 3 passengers, as shown in Figure 7.9. Air was supplied through the diffuser inlet above the passenger heads and exhausted through the outlet near the floor. The ventilation rate was carefully set according to the ASHARE aviation standard (ASHRAE, 2009), which yielded a mass flow rate of 0.04 kg/s at the inlet for the 3-passenger cabin section. The air temperature at the inlet was 25 °C as recommended by the ASHRAE standard (ASHRAE, 2009). A fixed convective heat flux of 35.6 W was applied at the CTM surfaces. During the computations, the original CTM model and those generated from 20 and 30 simplification iterations were used, respectively. The numerical results yielded from the original CTM model were used as the baseline for comparison.

Figure 7.9: CFD model of the airliner cabin section.

The predicted airflow fields in Plane 1 (Figure 7.9) are shown in Figure 7.10. The results revealed that strong thermal buoyancy flows existed above the CTM heads, which were strong enough to push up the ventilation jet. However, the CTM simplification up to 30 iterations did not cause much change in the overall flow field, except in some small local regions close to the CTMs.
Figure 7.10: The predicted airflow fields in Plane 1.

Figure 7.11 presents a quantitative comparison of the velocity profiles along two horizontal lines in Plane 1. For Line 1 which was located 0.15 m above the CTM heads and was in the passengers’ thermal plume, an iteration number of CTM simplification up to 20 was still safe for a stable prediction of the airflow field. However, when the iteration number further increased up to 30, significant deviation appeared. Especially, for the predicted air velocity above Passenger C ($Y = 0.4m$), the local error could be over 30%. On the contrary, for Line 2 which was located near the baggage compartment and was outside of the thermal plume, the predicted velocity profile was constant even the iteration number reached up to 30.

Figure 7.11: Velocity profiles along lines in Plane 1.

Therefore, the simplified CTMs, although they did not significantly influence the predicted airflow field outside the thermally affected regions, they had a direct effect
on predicted the thermal plumes. Despite the limited distance range from the CTM surface, the inaccuracy in airflow field prediction would enlarge the errors associated with contaminant transport. This is especially true when the contaminants are released from human bodies (e.g. pathogen-carrying dust/droplets or VOC). In order to demonstrate this point, further computations were conducted. In the computations, it was assumed that volatile aerosols generated from the ozone reaction with human skin lipid (Wang and Waring, 2014; Rai et al., 2015) were evenly released from skin of the passenger close to the aisle (Passenger C).

The contour plots of predicted aerosol concentration are shown in Figure 7.12. For the convenience of comparison, the concentration was normalized based on the maximum concentration in the cabin section. Compared with Figure 7.10, the CTM simplification had more significant effect on the modelling of contaminant transport than the air velocity prediction when the iteration number exceeded 20. As shown in 7.12c, significantly different contour patterns were observed in the area above the passenger heads from those in Figure 7.12a and 7.12b.

Figure 7.12: Predicted squalene concentration in Plane 1.

Figure 7.13 shows a quantitative comparison of the normalized contaminant concentration in the passenger breathing zones. The concentration profiles were plotted along the horizontal line starting from the nose (see Figure 6 for the exact location) of each passenger. According to Figure 7.13, the error of contaminant concentration prediction in the breathing zone caused by the CTM simplification could be different depending on the local airflow conditions. For passenger C who was seated in the region with stronger thermal buoyancy flow, the error was enlarged with further CTM simplification. On the other hand, for Passenger A who was seated close to the window where the airflow was comparatively weak, the predicted contaminant
concentration was less sensitive to the CTM simplification. However, for anyone of
the three passengers, the predicted squalene concentration was quite stable when the
iteration number was no larger than 20. Therefore, it could be concluded that even
for such a densely occupied indoor space with forced ventilation scheme, the CTM
simplification approach proposed in this study is still safe when the iteration number
does not exceed 20, or the simplification index is not larger than $3.5 \times 10^{-4}$.

![Figure 7.13: Normalized squalene concentration in the passenger breathing zones.](image)

### 7.1.4 Conclusions

While simplified computational thermal manikins (CTMs) could largely reduce the
computational cost of CFD simulations of occupied indoor spaces, they also cause
inaccuracy in the numerical results. On the contrary, 3D-scanned manikin models
with detailed body features are capable of contributing to an improved accuracy,
however, they are generally found to pull down the computational efficiency. In this
study, a CTM simplification approach based on the mesh decimating algorithm of
Garland and Heckbert (1997) was used to simplify the 3D-scanned manikin models
for the purpose of reducing the computational cost while maintaining the predictive
accuracy. CFD computations were conducted using gradually simplified CTM models
generated from the approach, in both sparsely and densely occupied indoor spaces.
The predicted fields of airflow and contaminant concentration were analyzed and
compared against each other. Conclusions arising from this study are as follows:

1. The CTM simplification approach provides a quantifiable, controllable and cost-
efficient way to simplify 3D-scanned CTM models, which allows a flexible sim-
plification control based on the given conditions. In addition, the simplified CTMs allow coarser mesh for CFD computations, without compromising the numerical stability and accuracy, which further improves the computational efficiency.

2. Although the effect of CTM simplification on the prediction of airflow field was mainly detected in the occupants’ thermally-affected regions, it could cause significant inaccuracy in the contaminant concentration field in the whole domain. However, through controlling the iteration number or the simplification index of CTM simplification, the accuracy of using a simplified CTM is still comparable to that of using a 3D-scanned manikin model. The computations demonstrated that even for a densely occupied indoor space such an airliner cabin, an iteration number of 20 or a SI less than $3.5 \times 10^{-4}$ is still safe to maintain the accuracy.
7.2 Evaluations of Simplified CTMs on CFD Predictions of Thermal Flow and Contaminant Fields in Large Cabin Section

Abstract:

While simplified computational thermal manikins (CTMs) are widely employed in CFD modes of occupied indoor spaces in order to save the computational cost, a criterion of simplification is still absent and the effects of CTM simplification are yet not clear. In this study, six CTMs including a 3D scanned CTM and five simplified CTMs generated from various simplification approaches were employed to analyse the impact of CTM simplification on the prediction of airflow field and contaminant transport. Comparison of the predicted airflow field against the published data in the literature demonstrated that CTM simplification has a strong effect on the thermal airflow field prediction in the vicinity of manikin surfaces. For densely occupied indoor spaces such as a train cabin, the error induced by CTM simplification could be enlarged and further cause significant global error to the prediction of contaminant transport. This is especially true when contaminants are released from the CTMs. This study demonstrated that the mesh decimating algorithm is promising to simply CTMs that is not only able to reduce considerable computational cost but capable of maintain an acceptable predictive error.
7.2.1 Introduction

When designing the heating, ventilating and air conditioning (HVAC) system and assessing the indoor air quality (IAQ), the occupational comfort, health and safety are and will always be the most crucial criteria. In order to assess the thermal comfort and to estimate the health risks associated with contaminant exposures, thermal manikins (CTMs) representing the human occupants have been widely employed in the Computational Fluid Dynamics (CFD) investigations of various indoor and built environments (Gao and Niu, 2004; Taghinia et al., 2015). A wide variety of CTMs, varying from simple geometries such as a combination of cuboids, cylinders and etc. (Park et al., 2015) to 3D-scanned manikins with high-resolution body and facial features (Deevy et al., 2008), have been reported in the literature. Usually, CTMs with simple geometries require much lower computational cost by allowing coarser computational grid size. However, simple geometries may also lead to the loss of local airflow details near the CTM surfaces, even though the impact on the airflow in the bulk regions were reported to be less significant (Deevy and Gobeau, 2006). Detailed CTMs, on the contrary, are beneficial to improved predictive accuracy particularly in the near-skin regions but demanding relatively high computational cost. Due to the limitation of current computational capacity, detailed CTMs are usually used to analyse the thermal comfort and micro-environment of a single person (Sorensen and Voigt, 2003) without considering comprehensive surrounding effects, whereas simplified CTMs are widely employed to investigate the ventilation and contaminant transport and exposure in multi-occupant indoor spaces such as airliner and train cabins (Poussou et al., 2010).

Most existing studies treated human bodies as passive objects subjected to the environment when investigating the contaminant transport and exposure (Poussou et al., 2010; Isukapalli et al., 2013). However, in reality, human bodies are continuously in contact and interacting closely with their surroundings by serving as obstacles of ventilation airflow and exchanging heat and mass simultaneously (Melikov, 2015). Thus, human bodies should also be regarded as the prime heat source of thermal buoyancy flows in most modern built environments (Li et al., 2015c). Rim and Novoselac (2009) found that the thermal plume induced by human body metabolic heat plays an important role in transporting contaminants from near-the-floor level into the breathing zone. This uprising thermal buoyancy flow was even found to be responsible for the connection between skin disease and respiratory disease (Lewis et al., 1969). In addition, taught from the lessons of global outbreaks of transmissible diseases such as SARS and H1N1 flu, the non-linear transport and exhalation-inhalation
characteristics of pathogen-carrying droplets in densely occupied indoor environment (e.g. airliner and train cabins) have become a major research concern (Sze To et al., 2009). More recently, it was reported that the ultra-fine particles and semi-volatile organic compounds yielded from the ozone reactions with human skin lipids (Gao et al., 2015) have been identified as long-ignored but serious health threats to office occupants, airliner passengers and metro commuters (Zhang and Chen, 2007a). Therefore, it is essential to consider human bodies as not only the sink but the source of the contaminants.

For CFD simulations with the aforementioned scenarios where contaminants are release from human bodies, CTMs with detailed body features are preferred in order to effectively capture the airflow field in the vicinity of human bodies and its effects on the overall contaminant distribution. However, this is apparently unpractical for multi-occupant spaces such as airliner and train cabins, in which a large number of CTMs (over 200) are involved. Therefore, compromise has to be made in order to balance the accuracy against the cost. It seems that more attentions were paid on the overall airflow and contaminant distributions whereas compromises were mostly made on the human bodies due to the complexity of body features (Park et al., 2015; Horikiri et al., 2015). However, a criterion about how to simplify CTMs for a multi-occupant space is still absent and thereby some CTM simplifications for multi-occupants simulations reported in the literature were quite arbitrary (Rai and Chen, 2012). Through using different CTMs in their CFD models, Mazumdar et al. (2011) proved that the predicted contaminant concentration field in an airliner cabin section was strongly affected by the CTM geometry. Therefore, in order to effectively predict contaminant transport in multi-occupant spaces and to expedite the understanding of the interactions between human occupants and their surroundings through CFD simulations, it is crucial to develop the CTMs properly with certain criteria. This is particularly important when contaminants are released from the occupants.

In this study, several CTM simplification approaches reported in the literature were employed to develop CTMs for CFD simulations and compared against each other. In order to address the strong effect of CTM geometry in the near-skin regions and to find the optimal approach, this study started with a single CTM seated in quiescent air. Then, further computations were conducted using a fully-occupied high-speed train cabin section to analyse the effects of CTM simplification on the prediction of contaminant transport. This study demonstrated that the CTM simplification approach based on the mesh decimating algorithm is promising to simplify multi-occupant indoor space models while remaining the desired predictive accuracy.
7.2.2 Methods

7.2.2.1 The computational thermal manikins (CTMs)

A 3D-scanned female manikin model with realistic human body features was selected as the original and baseline computational thermal manikin (CTM-1) in this study, as illustrated in Figure 7.14a. This model has been previously used in several studies due to its detailed human body features (Sorensen and Voigt, 2003; Li et al., 2015c). More information of this model is available in the open database (http://www.ie.dtu.dk/manikin). This original model was then simplified with the criteria or approaches reported by other researchers and five CTM models were thereby developed.

CTM-2 was developed mainly by manually smoothing and simplifying the original model (CTM-1), as shown in Figure 7.14b. Most of the body features were remained, while some extremely complicated facial features from CTM-1 were eliminated, such as the ears, eyes and the mouth. These facial features would require extremely fine mesh element and significant number of mesh elements but could have very limited contributions to the simulations. Eliminating these features would directly reduce the total number of mesh elements, while most of the body features (e.g. upper torso, arms and legs) would be able to remain unchanged.

A mesh decimating algorithm (Garland and Heckbert, 1997) was then employed to simplify the original model as the second approach without removing or reconstructing body features. By applying the algorithm, very complicated 3D geometry surfaces were initially divided into a number of triangular faces which are fine enough to capture all the body features. Then, the number of these triangular faces was gradually reduced by merging faces with similar curvatures or replacing with larger triangular faces. Thus, by using this approach, the outlines of all body features would be able to be preserved, while only the fidelity of the model is reduced. According to Li et al. (2015c), the original model (CTM-1) was firstly divided into 250,000 initial triangular faces, followed by further reductions of the triangular face numbers with a number of iteration steps. The steps of iteration are controlled and the degree of triangular face contraction was set to be uniform for each iteration step (Li et al., 2015c). As demonstrated in Figure 7.14c and 7.14d, CTM-3 and CTM-4 were developed based on this approach by simplifying CMT-1 with different reduction levels of the triangle faces. In order to quantify the mesh decimating algorithm, custom iterations were set to easily distinguish the different levels of simplifications. Therefore, CTM-3 and
Figure 7.14: Geometric information of CTMs.

- **CTM-1**
  - Original model
  - Commonly used model (Sorensen and Voigt 2003, Li et al. 2015)

- **CTM-2**
  - Unnecessary facial features were eliminated

- **CTM-3**
  - Surface smoothed model
  - 15 iterations

- **CTM-4**
  - Surface smoothed model
  - 30 iterations

- **CTM-5**
  - Skeleton based rebuilt model
  - Much simpler body features

- **CTM-6**
  - Surface area and key dimensions based model
  - Extremely simple body features
CTM-4 can be considered as being simplified by 15 iteration steps and 30 iteration steps, respectively.

CTM-5 was a completely rebuilt manikin model based on the key skeleton structures of the original model (CTM-1), as recommended by Ruzic and Bikic (2014). The key skeleton structures of the original model were firstly extracted and carefully measured. Then the new CTM was rebuilt by referring to the exactly same skeleton structures but using much simpler surfaces to describe body features. From Figure 7.14e, it can be noticed that only basic features (e.g. nose, elbow, waist and etc.) were contained in CTM-5.

Lastly, an extremely simplified manikin model (CTM-6) was created with very limited body features according to the body surface area (BSA) and the key dimensions of the original model. This simplification approach was initially mentioned by Miyanaga et al. (2001) and then has been widely used by other researchers (Zhang and Chen, 2007a; Rai and Chen, 2012) under multi-occupants and very complex indoor environment such as airliner cabins. By using this approach, the upper torso of CTM-6 was consisting of very simple cylinders only, while the lower body was built by simple blocks, as demonstrated in 7.14f.

All the manikin models were divided into several body segments (e.g. head, torso, arms and etc.) for the purpose of comparing the weighting factor of each segment, as listed in Table 7.2. Detailed comparisons of the overall BSAs and the surface areas of each individual body segment among all the aforementioned CTM models were also given in Table 7.2. Since the distribution of the applied heat load would be directly related to the BSA of each body segments, the weighting factors of each body segment on the simplified CTMs were controlled to be close to the original model (CTM-1). Thus, the distribution of the heat load would be similar among all the CTMs. The overall BSA was 1.596 $m^2$ of the original model (CTM-1) and ranged from 1.530 $m^2$ to 1.638 $m^2$ for the rest CTMs, which mostly agreed well with the statistical mean female BSA (1.522 $m^2$) from Yu et al. (2010) anthropometric data and the numerical model by Topp et al. (2002).
Table 7.2: Body surface areas and segment weighting factors of CTM models.

<table>
<thead>
<tr>
<th></th>
<th>CTM-1</th>
<th></th>
<th>CTM-2</th>
<th></th>
<th>CTM-3</th>
<th></th>
<th>CTM-4</th>
<th></th>
<th>CTM-5</th>
<th></th>
<th>CTM-6</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Weighting factor (m²)</td>
<td>Weighting factor (%)</td>
<td>Weighting factor (m²)</td>
<td>Weighting factor (%)</td>
<td>Weighting factor (m²)</td>
<td>Weighting factor (%)</td>
<td>Weighting factor (m²)</td>
<td>Weighting factor (%)</td>
<td>Weighting factor (m²)</td>
<td>Weighting factor (%)</td>
<td></td>
</tr>
<tr>
<td>HEAD</td>
<td>0.101</td>
<td>6.3</td>
<td>0.097</td>
<td>6.2</td>
<td>0.100</td>
<td>6.3</td>
<td>0.097</td>
<td>6.3</td>
<td>0.096</td>
<td>6.1</td>
<td>0.122</td>
</tr>
<tr>
<td>NECK</td>
<td>0.022</td>
<td>1.4</td>
<td>0.022</td>
<td>1.4</td>
<td>0.021</td>
<td>1.4</td>
<td>0.021</td>
<td>1.4</td>
<td>0.022</td>
<td>1.4</td>
<td></td>
</tr>
<tr>
<td>UPPER BODY</td>
<td>0.450</td>
<td>28.2</td>
<td>0.448</td>
<td>28.6</td>
<td>0.445</td>
<td>28.2</td>
<td>0.432</td>
<td>28.2</td>
<td>0.482</td>
<td>30.5</td>
<td></td>
</tr>
<tr>
<td>Arm L</td>
<td>0.159</td>
<td>10.0</td>
<td>0.151</td>
<td>9.7</td>
<td>0.157</td>
<td>10.0</td>
<td>0.152</td>
<td>10.0</td>
<td>0.157</td>
<td>9.9</td>
<td>0.798</td>
</tr>
<tr>
<td>Arm R</td>
<td>0.160</td>
<td>10.0</td>
<td>0.151</td>
<td>9.7</td>
<td>0.158</td>
<td>10.0</td>
<td>0.153</td>
<td>10.0</td>
<td>0.158</td>
<td>10.0</td>
<td></td>
</tr>
<tr>
<td>Leg L</td>
<td>0.352</td>
<td>22.1</td>
<td>0.349</td>
<td>22.3</td>
<td>0.348</td>
<td>22.1</td>
<td>0.337</td>
<td>22.1</td>
<td>0.333</td>
<td>21.1</td>
<td>0.359</td>
</tr>
<tr>
<td>Leg R</td>
<td>0.351</td>
<td>22.0</td>
<td>0.348</td>
<td>22.2</td>
<td>0.347</td>
<td>22.0</td>
<td>0.337</td>
<td>22.0</td>
<td>0.333</td>
<td>21.1</td>
<td>0.359</td>
</tr>
<tr>
<td>Total area</td>
<td>1.596</td>
<td></td>
<td>1.566</td>
<td></td>
<td>1.577</td>
<td></td>
<td>1.530</td>
<td></td>
<td>1.580</td>
<td></td>
<td>1.638</td>
</tr>
</tbody>
</table>

7.2.2.2 CFD computations and boundary conditions

The experimental measurements from Licina et al. [24] were utilised in this study for the purpose of validating and comparing the numerical results. In the experiment, a sitting manikin with 15° back-leaning posture was placed in a large enclosed chamber with dimensions of $11.1 \times 8 \times 2.6 \text{ m}^3$. Quiescent condition was achieved by turning off all the vents during the measurements. The sitting manikin model was constantly heated and the ambient temperature was maintained at 26 °C. The airflow was exclusively driven by the thermal plume generated by the body heat. The airflow and temperature were captured using the Particle Image Velocimetry (PIV) technique and complemented with the Pseudo Colour Visualisation (PCV) technique in their study. The measured data was compared against the numerical results in this study.

Since the chamber environment was controlled to be quiescent during the experiment, numerically, it is possible to meet the quiescent condition by applying a smaller computational domain. The computational domain with dimensions of $4 \times 3 \times 2.6 \text{ m}^3$ was proved to be sufficient to meet the requirement (Rim and Novoselac, 2009), as illustrated in Figure 2. Free-flow openings with zero gauge pressure to allow air flowing in and out freely were set at the front, back and side walls of the domain. The original model CTM-1 (Figure 7.15a) was initially adjusted to match the same sitting posture as the one used in Licina et al.’s experimental (Figure 7.17b). All the simplified CTMs were developed based on the adjusted CTM-1. A total heat loss of 89 W was measured from the nude manikin body in Licina et al. (2014)’s experiment, in which the convective heat loss was approximately 40% of the total heat loss (Sorensen and Voigt, 2003; Myrakami et al., 2000). Thus, a convective heat load of 35.6 W was applied at all CTMs, while the radiation was not considered in this study.
Also, the segmental Bio-heat and segmental skin variation (Antoun et al., 2016) was not considered in this study due to the fact that the weight of each body segment would be different among the CTMs through different simplification approaches, especially for CTM-6 which contains less body segments than the other CTMs. The convective heat load and heat flux of each CTM were listed in Table 7.3, which were very close to the published experimental measurements (Licina et al., 2014; deDear et al., 1997) and numerical results (Li et al., 2015c; Topp et al., 2002).

![Figure 7.15: Computational domain; (a) CTM-1; (b) Manikin model by (Licina et al., 2014).](image)

The computational domain was discretised using unstructured tetrahedron mesh. Mesh size applied on the body surfaces of each CTM varied depending on the level of simplification, while relatively coarse mesh elements were used at the bulk region and controlled to be uniform in all cases. Each case was computed with three different mesh configurations and tested through checking the mesh quality in ICEM 16.0 (ANSYS, 2015), the grid convergence index (GCI) (Roache, 1994) and the $y+$ values at the manikin surfaces (ANSYS, 2015; Habchi et al., 2016). For the CTM-1 case, the manikin body surface was computed by the mesh size of 5 \( mm \), 10 \( mm \) and 20 \( mm \), respectively. Ten prism layers adjacent to the manikin surface with total height of 15 \( mm \) were generated to improve the boundary layer resolution. The mesh independence of CTM-1 was achieved at 2.4 million with the surface mesh.
size of 10 mm, the overall mesh quality above 0.4 and the maximum y+ values at the manikin surface less than 3. Same criteria were utilised to conduct the tests on the other cases, in which relatively coarser meshes were applied on the CTMs with higher degree of simplification and vice versa. Thus, the total mesh elements of all studied single CTM cases varied from 1.9 million to 2.4 million. The airflow field inside the computational domain was solved using the incompressible Navier-Stokes equations with the Boussinesq approximation for the thermal buoyancy flows. The model equations were discretised using the conservative finite-volume method and the SIMPLEC algorithm was employed for the velocity-pressure coupling.

Table 7.3: Comparison of boundary conditions on the manikin bodies.

<table>
<thead>
<tr>
<th>CTM</th>
<th>Body surface area (m²)</th>
<th>Total heat load (W)</th>
<th>Convective heat load (W)</th>
<th>Convective heat flux (W/m²)</th>
<th>$\Delta T_{max}$ (°C)</th>
<th>Convective heat transfer coeff (W/m²°C)</th>
</tr>
</thead>
<tbody>
<tr>
<td>CTM-1</td>
<td>1.59</td>
<td>89</td>
<td>35.6</td>
<td>22.39</td>
<td>5.39</td>
<td>3.63</td>
</tr>
<tr>
<td>CTM-2</td>
<td>1.57</td>
<td>89</td>
<td>35.6</td>
<td>22.68</td>
<td>5.29</td>
<td>3.61</td>
</tr>
<tr>
<td>CTM-3</td>
<td>1.58</td>
<td>89</td>
<td>35.6</td>
<td>22.53</td>
<td>5.40</td>
<td>3.63</td>
</tr>
<tr>
<td>CTM-4</td>
<td>1.53</td>
<td>89</td>
<td>35.6</td>
<td>23.27</td>
<td>5.61</td>
<td>3.66</td>
</tr>
<tr>
<td>CTM-5</td>
<td>1.58</td>
<td>89</td>
<td>35.6</td>
<td>22.53</td>
<td>4.42</td>
<td>3.45</td>
</tr>
<tr>
<td>CTM-6</td>
<td>1.63</td>
<td>89</td>
<td>35.6</td>
<td>21.84</td>
<td>4.87</td>
<td>3.54</td>
</tr>
<tr>
<td>deGare et al. 1997 (Exp.)</td>
<td>1.47</td>
<td>NG</td>
<td>33.98</td>
<td>23.1</td>
<td>7</td>
<td>3.3</td>
</tr>
<tr>
<td>Licina et al. 2014 (Exp.)</td>
<td>NG</td>
<td>89</td>
<td>NG</td>
<td>NG</td>
<td>6</td>
<td>3.78</td>
</tr>
<tr>
<td>Topp et al. 2002 (CFD)</td>
<td>1.52-1.61</td>
<td>NG</td>
<td>38</td>
<td>23.60-25.00</td>
<td>5.1</td>
<td>5.0-7.4</td>
</tr>
<tr>
<td>Li et al. 2015 (CFD)</td>
<td>1.59</td>
<td>89</td>
<td>35.6</td>
<td>22.39</td>
<td>5.39</td>
<td>3.63</td>
</tr>
</tbody>
</table>

In terms of the contaminants transport and distributions, the human body has been treated as the source of the gaseous contaminants (Zhang and Chen, 2007a; Rai and Chen, 2012). Chemical reaction between the human skins lipids and the surrounding secondary organic aerosols (SOAs) (e.g. ozone) have been proved as an important source of producing gaseous contaminants such as volatile organic compounds (VOCs) and ozone reactions (Gao et al., 2015). By interacting with the buoyancy driven thermal plume, gaseous contaminants concentration could be potentially lifter up and suspend in the occupants’ breathing zones.
Therefore, in this study, the gaseous contaminants were assumed to be released evenly from the CTM surfaces (body skins). Contaminant transport was solved using a transportable scalar (ANSYS, 2015),

\[
\frac{\partial}{\partial t}(\rho C) + \nabla \cdot (\rho(C\vec{U} - D\nabla C)) = S_C
\]

(7.9)

where \(C\) is a scalar representing the contaminant concentration, \(S_C\) is the source term, \(D\) is the kinematic diffusivity of the contaminant and can be predicted by (Cussler, 2009):

\[
D = \frac{1.858 \times 10^{-3} T^{3/2}}{p \sigma_{ij} \Omega} \sqrt{\frac{1}{M_i} + \frac{1}{M_j}}
\]

(7.10)

where \(i\) and \(j\) are the molecules denotations in the gaseous mixture, \(T\) is the absolute temperature, \(M\) is the molar mass, \(p\) is the pressure, \(\sigma_{ij}\) is the average collision diameter, and \(\Omega\) is a temperature-dependent collision integral.

Oxidation products such as acetone, 6-methyl-5-hepten-2-one (6-MHO), geranyl acetone and hexanal can be generated through the chemical reactions between ozone and human skin lipids, in which squalene, linoleic acid (LA) and oleic acid (OA) were the main precursors (Zeng et al., 2013; Thornberry and Abbatt, 2004). The molar mass of these oxidation products ranges from 58.08 g/mol to 196.29 g/mol with the average molar mass of 120 g/mol. The corresponding kinematic diffusivity in the air was set as \(12.8 \times 10^{-4} m^2/s\).

The aforementioned model equations were solved by the commercial CFD code CFX 16.0 (ANSYS, 2015). Steady-state computations were conducted in conjunction with the RNG k-\(\varepsilon\) model for the air turbulence due to its successful utilisation in simulating dilute multiphase flows under thermal environments around human bodies (Tu and Fletcher, 1994; Gao and Niu, 2005).

### 7.2.3 Results and discussion

#### 7.2.3.1 Thermal airflow field

Under a quiescent environment, the airflow around the manikin body was dominated by the thermal plume due to the temperature difference between the body surface and the ambient air. It is essential to check the natural convective heat transfer rate of all the CTMs and thereby the convective heat transfer coefficient \((h_c)\) was selected as a criterion. According to (Gao and Niu, 2004; deDear et al., 1997), a commonly used
approximation of the $h_c$ for the human body under natural convection was applied and can be calculated by:

$$h_c = 2.38(t_{\text{skin}} - t_{\text{atm}})^{0.25} W/(m^2 C) \tag{7.11}$$

The $h_c$ values of all CTMs were calculated and compared to the published data, as listed in Table 7.3. The convective heat transfer coefficient of all the studied CTMs ranged from 3.45 to 3.63 $W/m^2 C$, which agreed well with the experimental measurements reported in the literature, as listed in Table 7.3.

The vertical plane across the centre of the manikin body ($Y = 0 m$) was selected to present the overall airflow field, illustrated in Figure 7.16. With the sitting posture, the predicted results revealed two significant thermal buoyance flows above the main torso and the legs, respectively, which were also captured in Licina et al. (2014)’s experiment. As the heat load was applied evenly to the manikin body, the thermal plume induced by the torso with larger BSAs was more significant and intense than the others. The maximum velocity above the manikin head was around 0.25 m/s, which agreed well with the experimental measurements in the literature (Craven and Settles, 2006; Salzmanzadeh et al., 2012).

![Figure 7.16: Airflow field in the Y = 0 m plane (CTM-1).](image)

The predicted thermal airflow profiles were further compared to the experimental measurements and previous numerical results at some selected positions in the vicinity of the CTMs. The velocity and temperature profiles were plotted at ten local points with various heights but all 3cm offset from manikin surfaces, as illustrated in Figure 153.
7.17a and 7.17b respectively. The original model (CTM-1) accurately predicted both the velocity and temperature profiles that agreed very well with the published results with maximum deviation of 4.2%, despite the slight deviation at the near floor points, which were supposed to be caused by the position difference of the manikin legs. For the simplified model by the mesh decimating algorithm, simulations using CTM-3 successfully predicted very close velocity and temperature distributions to the published results with errors less than 5% and 1%, respectively. However, a noticeable deviation appeared when the iteration steps increased to 30 (CTM-4), especially for the temperature field (deviation in excess of 8%). Simulations with the other CTMs failed to capture the velocity or temperature profiles, although the CTM-2 case was able to capture similar velocity and temperature patterns at the selected points.

![Velocity and Temperature Profiles](image)

(a) Velocity profiles.  
(b) Temperature profiles.

Figure 7.17: Comparison of velocity and temperature profiles at selected points.

The body feature difference of thermal manikins could directly affect the airflow field in the vicinity of the CTMs and its impact would be enlarged in the upper regions (e.g. breathing zone and the region above manikin head) due to the development of the buoyancy driven thermal plume. Since the air quality inside human’s breathing zone is one of the major criteria when designing or assessing the HVAC system in indoor environments (Sekhar, 2015), the effect of the manikin geometric differences was analysed in the breathing zone of each CTMs. The predicted airflow fields were compared to the published experimental data and numerical results (Li et al., 2015c; Licina et al., 2014), as shown in Figure 7.18. By using original manikin model (CTM-1) with full body features, airflow prediction achieved from CTM-1 case was very close
to the published experimental and numerical results. The good agreements of the comparisons in the breathing zone between the CTM-1 case and the published data demonstrated that the applied numerical settings were reliable to further predict and describe the airflow pattern change in the micro-environment affected by the body geometries.

![Figure 7.18: Comparison of velocity vectors at breathing zone.](image)

It can be noticed from Figure 7.18d and 7.18e that by using reasonably simplified manikin models (CTM-2 and 3), the predicted airflow patterns in the breathing zone were still very similar to the published data and results. However, the velocity fields failed to be captured when the body features were over-simplified (CTM-4 and 5), especially at the facial region very close to the surface of manikins. For CTM-6 case, the error of predicted airflow profiles was further enlarged. A significant low velocity region can be clearly observed in Figure 7.18h which was obviously abnormal and caused by excessively simplified body features.
In order to quantitatively investigate the effect of CTM simplifications, the velocity profiles were compared at selected positions in the breathing zone of CTMs. A horizontal line (just in front of the nostrils) with a length of 150 mm was selected to capture the velocity change from the manikin surface to its micro-environment, as demonstrated in Figure 7.19a. Again, the velocity magnitudes predicted by the CTM-1 (original model) case and the CTM-3 (15 iterations) case agreed very well with the experimental measurements. Both models predicted very close vertical velocity distributions to the experimental data with maximum error less than 2%. Results from CTM-2 (facial features eliminated) case still captured similar airflow pattern, whilst the horizontal velocity profiles were significantly smaller than the published data with maximum deviation greater than 7%. The error yielded using CTM-6 was further enlarged to be more than 10% for both horizontal and vertical velocity predictions. Similar findings can be also observed in Figure 7.19b, which plotted the velocity profiles at 10 select points with same offset in the breathing zone only.

By comparing the numerical results among the original model case, published numerical results and experiment measurements, the reliability of the numerical settings and the applied approach was verified based on the good agreements. The average predictive error was under 3% for the CTM-1 case. In terms of the model simplifications, the comparisons of airflow field in the micro-environment around manikin bodies indicated that both CTM-2 (facial features eliminated approach) and CTM-3
(mesh decimating algorithm approach with 15 iterations) would be able to predict similar velocity profiles at the micro-environment around manikin bodies. However, when the focus was on the breathing zone, the CTM-3 with the mesh decimating algorithm approach performed better than the facial features eliminated model (CTM-2) in relation to the airflow profiles.

7.2.3.2 Contaminant transport

In this study, the manikin body was considered as the source of the contaminant. The numerical results from the CTM-1 case were firstly given to illustrate the global contaminants distribution at two selected plane (X = 1.8 m and Y = 0 m), as shown in Figure 7.20. The contaminants were clearly dominated and driven by the ascending thermal plume and almost followed the same pattern as the airflow field (Figure 7.16), especially at the region above the manikin head and shoulders. The concentration level was very low at the bulk regions far away from the manikin body, whereas higher concentration can be observed in the vicinity of the upper torso and above the manikin head.

![Figure 7.20: Global distribution of the contaminants released by CTM-1.](a) Front view at X=1.8m plane. (b) Side view at Y=0m plane.)
In order to describe the contaminants distribution in the vicinity of the CTMs, the normalised contaminants concentrations were plotted along three lines. As illustrated in Figure 7.21, Line 1 was placed in front of the manikin with the same tilted angle as the sitting posture, while a horizontal line in front of the manikin head and a longitudinal line above the manikin head were selected as Line 2 and Line 3, respectively. Among these three plots, the most significant deviation on the contaminants concentration caused by the body difference occurred at Line 1 (Figure 7.21a). This was mainly due to the fact that Line 1 was placed very closely to the manikin body and almost the entire line was inside the thermal plume affected region. The difference between the concentration profiles predicted using different CTMs became more obvious at the upper torso of CTMs (Z > 0.6 m), in which the effect of the buoyancy driven thermal plume was also maximised. The results from Line 2 and 3...
revealed that the effect of the thermal plume on the contaminants distribution was considerable in the thermally affected regions and eventually caused very high concentration level in the regions just in front of the manikin face (Line 2 at \( Y = 0 \) m) and above the manikin head (Line 3 at \( X = 1.7 \) m), as illustrated in Figure 7.21b and 7.21c.

When considering the effect of CTM simplifications, the mesh algorithm based model with 15 iterations (CTM-3) successfully predicted very close concentration profiles to the original model (CTM-1) at all selected lines with maximum deviation less than 3%. By eliminating some facial features, CTM-2 also obtained very good agreement on the concentration profiles to the CTM-1 at Line 2 and Line 3. However, noticeable deviations (more than 10%) on the concentration profiles between CTM-1 and 2 can be observed at some regions of Line 1, although the predicted pattern from the CTM-2 case was still similar to the original model. On the other hand, when the model was overly simplified (i.e. CTM-5 and 6), the predictive errors became more significant at all plots. Overall, the contaminants concentration was considerably high in the regions in front of the manikin body and above the manikin head, while the main prediction errors on the contaminants concentration occurred in the same regions. Therefore, more attentions should be drawn at those corresponding regions (i.e. breathing zone and the region atop the manikin head).

The concentration of the contaminants was further compared at human’s breathing zone and the region above the manikin head based on the aforementioned findings. A square plane \( 0.36 \times 0.6 \) m was placed at the centre \( Y = 0 \) m of the emphasised region. The normalised concentration contours were shown in Figure 7.22. The contour results indicated that the buoyancy driven thermal plume had significant effect on and mainly dominated the contaminants distribution, while the highest concentration occurred just atop the skull. The contaminants released from the lower body segments were carried upward by the ascending thermal plume and constantly distributed in front of the nose and mouth. Since the effect of thermal plume was significant enough to bring considerable amount of the contaminants into human’s breathing zone, proper design of the indoor HVAC setup is strongly required to avoid near-floor level contaminants being carried upward and inhaled by the occupants.

In terms of the model simplifications, the concentration contour predicted by CTM-1 (Figure 7.22a) was applied as the baseline to compare with the simplified models. It can be found from Figure 7.22b to 7.22d that most simplified models were be able to predict similar contaminants distributions at the interested regions. However, when investigating the thickness of each contour layers, only the CTM-3
case successfully predicted the same results as the original model (CTM-1), while CTM-2 obtained similar results at higher concentration regions but much thinner contour layer at lower concentration regions. The CTM-6 case, on the contrary, failed to predict consistent results with the original model, which proved that using manikin models with very limited body features (i.e., using simple blocks or cylinders) would not be able to predict accurate contaminants transport and distribution when human bodies are considered as the source of the contaminants.

Figure 7.22: Concentration contour at Plane 1 in the breathing zone.
Figure 7.23: Concentration contour at Plane 2 above the manikin head.

Also, a second square plane (plane 2) with the same size as the plane 1 was placed 0.3 m above the manikin head to investigate the contaminant distribution and concentration at the maximum local air velocity region atop the manikin, as illustrated in Figure 7.23. The numerical results indicated that the contaminants distributed much wider atop the CTMs due to the acceleration and dispersion of the thermal plume. A large area of distributed contaminants with considerable concentration can be observed from X = 1.7 m to X = 2.0 m (the main torso of the manikin was
placed at X = 1.7 m along the longitudinal direction. Contaminants concentrated inside this region were brought up from the lower height level which further revealed a considerable high contaminants concentration at the similar but relatively smaller regions in front of the manikin (i.e. breathing zone).

The patterns of the concentration contours at plane 2 were also compared among all CTMs. Again, the CTM-3 case (Figure 7.23c) predicted the closest contaminants distribution and concentration to the original model. The CTM-2 case (Figure 7.23b) over-predicted the concentration contour, although the contour pattern was still similar. The sensitivity of the human body difference seemed to be higher at plane 2 than plane 1 and thereby led to more obvious deviations on the contour patterns for the rest of the CTMs. Therefore, the geometry difference of the CTMs would not only affect the local contaminants distributions but also cause the deviations at the micro-surroundings (thermally affected regions) around the CTMs.

7.2.3.3 Applications in cabin environment

Figure 7.24: Future applications of tested CTMs (CTM-3 & CTM-4) in 4-rows train cabin.

When the indoor spaces become more complicated with increased number of occupants, the computational cost will be significantly increased and simplifications of the geometry models will be strongly required to make the simulation feasible and efficient. On the other hand, if the CTMs are over-simplified using simple blocks or cylinders, which have been widely used in a number of published studies under multi-occupants and complex indoor environment (Zhang and Chen, 2007a; Wang et al., 2014a), the accumulated errors caused by each of the over-simplified CTMs would be significant. The conflict between the computational cost and the numerical
reliability seems to be hardly balanced in existing literature. However, it is possible to find the most proper simplification approach that can not only reduce significant computational cost but minimise the errors due to geometry differences. Based on the above outcomes of single CTM case in the enclosed chamber, the mesh decimating algorithm approach seemed to be more promising to meet this target.

![Velocity profiles at selected planes](image)

Figure 7.25: Velocity profiles at selected planes (15 cm in front of the sitting manikins).
A further test was conducted between two CTMs developed by the mesh decimating algorithm approach (i.e. CTM-3 & CTM-4) in a more complicated multi-occupants environment. A train cabin section containing four rows of passengers and seats was built based on the Chinese Railyway High-speed (CRH-2) (Wang et al., 2014a), as demonstrated in Figure 7.24. The ventilation parameter was set accordingly to the Chinese Railway stand (China, 1988). A total number of 24 CTMs were computed in this train cabin section. Through using CTM-3 models instead of the original CTMs, over 40% of the total mesh elements were saved, while a further 20% reduction on the computational cost was achieved when CTM-4 models were em-
ployed. All the boundary conditions were set to be the same between these two cases, except the CTMs. In terms of the airflow field, four vertical planes (one for each row) were selected 15 cm in front of the nose tips to analyse the velocity profiles, as illustrated in Figure 7.24. The velocity vectors and distributions predicted by the CTM-3 and CTM-4 models (Figure 7.25) were similar at the lower regions of most planes. However, at the regions in front of the main bodies and above the passengers’ heads, in which the effect of thermal plume was considerable, the airflow patterns were obviously different between these two cases. This was due to the deviations of the thermal plume patterns generated by different CTM, which were significantly enlarged when a large number of CTMs were placed very close to each other in a limited space.

For the contaminants transport and distribution, similar to the single manikin case, the gaseous contaminants were assumed to be released from the surface of passenger D sitting at the second row. Figure 7.26 provided a comparison of normalised contaminant concentration at the plane across Passenger D. High contaminant concentration can be noticed in front of and above Passenger D at row-2 in CTM-4 case, while the normalised concentrations at the regions far away from the source passenger did not show significant differences between two studied cases. In order to further visualise the impact of CTM simplifications, the contaminants distributions predicted by both CTM-3 and 4 cases at various configurations of iso-surface were compared in Figure 7.27. At the final low concentration state (iso-surface 5), it seems that both CTM 3 & 4 cases predicted similar results that the contaminants were widely dispersed into the whole cabin section and mainly affected the passengers sitting on the other side to the source passenger. However, when investigation was taken into details (with gradual reduced C* values), deviations of the contaminants distributions predicted using different CTMs became significant and can be clearly observed from iso-surfaces in Figure 7.28. The numerical results implied that the transport characteristics of contaminants with relatively high concentrations are very sensitive to the geometry of the CTMs when human skins are considered as the source of the contaminants, even though the contaminants distribution may be similar at the final state (low concentration). This is especially crucial when investigating the transport and distribution of infectious disease, since the transport characteristics of contaminants with considerable concentration would directly relate to the potential infected occupants.
Figure 7.27: Time dependent contaminants transport released by Passenger D at row-2.
Figure 7.28: Normalised contaminant concentrations at passengers’ breathing zones.

Since the potential health risks caused by the contaminants transport would be
closely related to the amount of the contaminants inhaled by occupants, the normalised contaminants concentration at passengers’ breathing zones were further compared in Figure 7.28. Passenger D sitting at row 2 was the sauce of the contaminants and the contaminants concentration at other passengers’ breathing zones were presented correspondingly to the passenger locations. The predicted results by CTM-3 Figure 7.28a showed relatively higher concentration levels at the breathing zones of passengers sitting behind (at row 3) the sauce passenger, while the highest amount of contaminants was predicted in the breathing zone of Passenger E sitting at the side of row 3. The CTM-4 case (Figure 7.28b) seems to predict a similar trend that the highest concentration occurred at the passenger sitting close to the side wall. However, the location of this passenger was one row forward than that predicted by CTM-3. This was probably caused by the deviations of the thermal plume development by the CTM simplifications as aforementioned. Based on the single CTM test, the ascending thermal plume generated by CTM-4 developed with less backward component above the head than the CTM-3. After placing a numerous number of CTMs very closely, the deviations on the thermal plume significantly enlarged and thereby affected the predictions of contaminants transport.

7.2.4 Conclusions

The thermal buoyancy flow and contaminant release from manikin sitting in a quiescent environment simulated. An original manikin model with full body details (CTM-1) and five simplified CTMs were applied to investigate the effect of body difference on the airflow and contaminants prediction in the vicinity of the manikin body and its micro-environment.

The thermal airflow field was firstly investigated by comparing the numerical results with the experimental measurements by Licina et al. (2014) and other published data (Li et al., 2015c). The original model (CTM-1), the simplified model based on the facial feature eliminated approach (CTM-2) and the simplified model with 15 iterations of the mesh algorithm (CTM-3) achieved satisfactory agreement to the compared data with maximum the error less than 3%. Other simplified models failed to obtain accurate thermal airflow profiles with reasonable errors due to the loss of main body features through simplifications.

Gaseous contaminants were assumed to be released from the manikin skin surfaces. The distribution of the contaminants was studied and compared among various CTMs. The contaminants released at lower height regions can be potentially brought upward by the buoyancy driven thermal plume. As a result, considerable
amount of contaminants were detected in the occupant’s breathing zone, while the highest contaminant concentration occurred atop the manikin head. The effect of the thermal plume would be maximised under multi-occupants environment and thereby cause much higher concentration of contaminants inside the occupants’ breathing zones and eventually lead to serious health hazards. Proper design of HVAC setups is strongly required and the personalised airflow device may be necessary to help avoiding contaminants suspending in the breathing zone.

In addition, CTM geometrical differences also caused significant deviations on the contaminant distributions at the micro-surroundings of the manikin body. The CTM-3 case achieved the best agreement to the original model with overall errors less than 3%, which means the mesh algorithm with proper iteration steps would have very limited impact on the numerical results (both on the airflow and contaminant fields). When the key focus was at the breathing zone, the facial feature elimination (CTM-2) seemed to have some impacts on the contaminant distribution (deviation greater than 10%), although the patterns could be very similar to the original model. Other CTMs with excessive degree of simplification were proved to be incapable of predicting accurate results.

Therefore, by testing the mesh algorithm based simplification approach under both single CTM and multi-occupants cases, this approach is recommended as an efficient and promising approach that can not only guarantee the accuracy of the numerical results, but effectively reduce the computational cost.
Chapter 8

Analysis of Airborne Disease Infection Risks in Fully-occupied Airliner Cabin

The main findings of this chapter have been published in:


In this paper, simulations were conducted using a Boeing 737 cabin model to study the transport characteristics of airborne droplets and the associated infection risks of passengers. The numerical results of the airflow field were firstly compared against the experimental data in the literature to validate the reliability of the simulations. Airborne droplets were assumed to be released by passengers through coughing and their transport characteristics were modelled using the Lagrangian approach. Numerical results found that the particle travel distance was very sensitive to the release locations, and the impact was more significant along the longitudinal and horizontal directions. Particles released by passengers sitting next to the windows could travel much further than the others. A quantifiable approach was then applied to assess the individual infection risks of passengers. The key particle transport information such as the particle residence time yielded from the Lagrangian tracking process was extracted and integrated into the Wells-Riley equation to estimate the risks of infection. Compared to the Eulerian-based approach, the Lagrangian-based approach presented in this study is more robust as it addresses both the particle concentration and particle residence time in the breathing zone of every individual passenger.
8.1 Introduction

After experiencing the fastest growth of passenger numbers in the past decade, there were more than 3.5 billion people travelling by air in 2015 and the number was forecasted to be more than doubled (7.4 billion) in 20 years (Tyler, 2015). Since the average occupancy of commercial flights was very high last year (around 80%) and is still increasing (Gill, 2016), the inflight conditions, such as air quality, thermal comfort and disease transmission risks have been drawing increasing attentions. Among these concerns, the transmission of airborne diseases is now in the spotlight, after the lessons taught from the global outbreaks of Tuberculosis (TB), Severe Acute Respiratory Syndrome (SARS) and Swine Influenza (H1N1) (Zhu et al., 2010).

To study the air quality and disease transmission in airliner cabins, a number of important affecting factors have been identified, such as human thermal plume (Yan et al., 2015) and passenger movements (Poussou et al., 2010). Among various types of contaminants found in airliner cabins, the infectious saliva/phlegm droplets released through coughing or sneezing has been emphasised in many epidemiology reports (Kenyon et al., 1996; Mangili and Gendreau, 2005). During the flight, since the passengers are sitting densely in a limited and enclosed space and unable to leave, diseases containing infectious pathogens (such as influenza and tuberculosis) released by index patients through coughing or sneezing would cause direct person-to-person infections (Escombe et al., 2007). Furthermore, the transmission of airborne diseases in airliner cabins revealed very strong non-linear characteristics in the investigations of several SARS infection cases in 2003 (Olsen et al., 2003), in which the relative locations of the infected passengers to the index patient were found very randomly distributed in the cabins. Therefore, as the perniciousness of the saliva/phlegm droplets has been widely raised, the knowledge of their transport behaviours in the cabin environment is crucial for precise predictions of the infection risks of every individual passenger.

To effectively assess the individual infection risk, epidemiology studies (Hass et al., 1999) in indoor spaces reported that two essential components are required: the intake dose of the infected person and the probability of infection under the estimated intake dose. A number of infection risk assessment models (e.g. the Von forester model (Fraser et al., 2004)) and the competing-risks model (Brookmeyer et al., 2004) were thereupon developed. Despite the diversity of the mathematical functions, infection risk assessment models can be summarised into two categories: deterministic models and stochastic models (Sze To and Chao, 2010). The deterministic models emphasised on the inherent tolerance dose of infectious person and infection only occur when the
intake dose of pathogens equivalent to or exceeding the tolerance dose, while the stochastic model mainly estimates the probability of acquiring the infection under the intake dose. Among various models, the Wells-Riley equation developed by Riley et al. (1978) based on Wells (1955)’s concept of quantum of infection, was widely used as the mathematical models in existing epidemic modeling (Escombe et al., 2007; Noakes et al., 2006), due to its universal applicability. In the Wells-Riley equation, the required threshold number of infectious airborne particles to cause infection can be defined as a quantum. Escombe et al. (2007) employed the Wells-Riley equation to predict the infection risks of tuberculosis under ventilated rooms in eight hospitals. The quantum concept was used to describe the infectious dose for tuberculosis in their investigation, which directly widen the applicable range of this model. The Wells-Riley infection risk assessment model offers a quick and flexible approach to assess the infection risks of different airborne diseases, which has been employed in a number of indoor space studies on infectious risks (Wu et al., 2016) and diseases transmission (e.g. tuberculosis transmission (Andrews et al., 2013; Nardell, 2016)). However, although commercial airliners had been determined as the major media carrying and transmitting the SARS worldwide in 2003 (Mangili and Gendreau, 2005), very few existing studies were conducted with attempts to predict the inflight infection risks of each passengers, due to the inherent complexity and particularity of the cabin environment.

When assessing the exposure dose related health risks in the cabin environment, existing studies mostly relied on the Eulerian-based concentration distribution of the droplets, to identify the high health hazard regions (Isukapalli et al., 2013). It is undoubtedly that the Eulerian-based approach can provide very fast 3D predictions of the contaminants concentration distribution, which is an important parameter when assessing the health risks because passengers sitting inside the high-concentration regions would usually have higher health risks. A most recent study conducted by You et al. (2017) employed the aforementioned Wells-Riley equation in conjunction with the Eulerian model to investigate the effect of the gaspers on the passengers’ exposure risks in a half-row cabin section. The Wells-Riley method was combined with the two-phase flow model when assessing the exposure risks in the cabin environment. In order to fit the Wells-Riley equation to the Eulerian model, they assumed that the exposure time was the same as the flight duration. However, the actual exposure time could be much less than the flight duration due to the cabin ventilation and is significantly different to every individual passenger, depending on the relative location to the index patient. The exposure time length in the Wells-Riley equation could be
a critical parameter affecting the infection risks. Beyond that, the particulate phase is assumed to be a continuum in the Eulerian framework, which directly leads to the loss of some critical information, such as the time of particle residence in a given domain. This shortage makes the Eulerian model physically untrue when assessing the infection risks, since the infectious pathogens are always released in conjunction with the droplets or particles and they are sharing the similar transport characteristics.

Alternatively, the Lagrangian particle tracking model was also utilised in several numerical studies (You et al., 2017; Gupta et al., 2011) due to its unique advantage in source-to-destination tracing of particle movement. Initial conditions of the released droplets/particles were also carefully in the existing studies. Gupta et al. (2011) numerically investigated the distribution of contaminants released through different behaviour (i.e. coughing, breathing and talking). They concluded that contaminants released by coughing of the index patient behaved similar as those from breathing, but the number is much higher. Chao et al. (2009) concluded that the geometric mean diameter of contaminants from coughing was $13.5 \mu m$ with average release speed of $11.7 m/s$. Although studies on initial conditions of the released particles are accumulating in the existing literature, investigations on the other key parameters (i.e. particle travelling distance and particle travelling time) were still inadequate in the cabin environment. Also, when providing detailed 3D characterised trajectories of the released particles, the Lagrangian model requires significantly high computational resources to track them. To save the computational cost, many studies (Ztek et al., 2010; Rai et al., 2013) used a reduced size of cabin section (3 rows or less) with unrealistic passenger models to imitate the cabin environment. Thus, the contaminants transport was significantly constrained by the computational domain and thereby the travel distance and time of contaminants could be misleading. Since airborne respiratory pathogens must reach the target infection site of the receptor to commence the infection, accurate predictions of the travelling distance and time of the infectious pathogens are crucial. Thus, it is necessary to apply an extended cabin section with adequate space and realistic passenger models with proper body features when assessing the transmission of airborne diseases. As a good start, Gupta et al. (2011) numerically investigated the transport of exhaled droplets in an extended seven-row cabin section. Their study provided detailed investigations on the droplets transport when the droplets were exhaled through coughing, breathing and talking. They found that the evaporation process happens very quickly (less than 0.3 second) and could be even faster with smaller particles. Since their study focused on the single exhalation behaviour of a passenger with evaporation process, the simulation was
extremely time consuming even after applying over-simplified manikin models. They also recommended other researchers to focus more on the effect of index passenger location on the expiratory droplet transport in the airliner cabins and to develop a quantifiable approach to assess infection risks. Later on, Gupta et al. (2012) further assessed the risk of influenza transport in the seven-row cabin with an index patient on board. They thoroughly introduced two approaches (i.e. the deterministic and probabilistic approaches) to calculate the influenza risk. The aforementioned Wells-Riley equation was employed in the probabilistic approach in their study to estimate the probability of influenza infection, which revealed a promising direction of assessing infection risks in airliner cabins. Their study was conducted under a twin-aisle cabin and they suggested further evaluations in relation to the passenger infection risks under different cabin layout and configuration. Since the focus in Gupta et al. (2012)’s study were the exploration and investigation of these two approaches, the diversity and thoroughness of the results that can be yielded by these approaches were unavoidably overlooked. Despite that, their studies have laid a pivotal foundation of assessing infections risks using a combined computational fluid dynamics (CFD) approach and the risk assessment model, although the passenger models were over-simplified as combination of regular blocks due to the extreme high computational cost on simulating contaminants transport in a seven-row cabin.

Therefore, with the awareness of using CFD related infection risks model has been established (Escombe et al., 2007; Gupta et al., 2012) and the increasing attentions of passenger infection risks in airliner cabins have been drawn (Gupta et al., 2011), this study further and carefully evaluated the infection risks of every individual passenger in a single-aisle airliner cabin section and contributed a systematic approach to analyse the infection risks in cabin environments using a validated CFD model in conjunction with the quantifiable risk assessment model. A seven-row cabin model based on the Boeing 737 was utilised in conjunction with 42 validated manikin models to imitate a more realistic cabin environment. Particulate contaminants were released through coughing by different passengers and tracked using the Lagrangian tracking model. The concentration distribution of contaminants was obtained by converting the particle trajectories using the so called particle source in cell (PSI-C) method (Zhang and Chen, 2007b). A quantifiable approach based on the Wells-Riley equation (Sze To and Chao, 2010) in conjunction with the Lagrangian model was applied to assess the infection risks in every passenger’s breathing zone. Diverse outcomes including both the transport trajectories and concentration distribution of the released contaminants, and the quantified infection risks of each passenger were yielded from
this study and thereby added important information to the current database in the literature in relation to the infection risks in the airliner cabins. Also, important parameters such as threshold number of infectious pathogens and the number of index patients are considered and controllable by this approach, which built an important guidance for further investigations of different diseases not only in the airliner cabins, but other densely occupied environments (e.g. high-speed rail and metro).

8.2 Method

8.2.1 Computational models

As one of the widely served medium-size commercial aircrafts during the SARS outbreak in 2003 (Mangili and Gendreau, 2005), Boeing 737-200 was referred as the prototype aircraft to develop the CFD cabin model and study disease transmission. A seven-row economy cabin section was numerically constructed with dimensions of 3.82 m × 2.15 m × 5.86 m (W×H×L), as illustrated in Figure 8.1, which contains 42 fully occupied passengers with 3-3 seat arrangement. The ventilation inlets and outlets were located at the upper and lower sides of the cabin walls, respectively. In terms of the computational thermal manikin (CTM) models, our previous study reviewed that proper body features of the manikin models are crucial for balancing the computational cost and accuracy (Yan et al., 2016). Thus, the simplified and validated CTM from our previous work was employed in this study as the passenger
model, as shown in Figure 8.2. Through contracting the pairs of triangle vertices, the key body features of the simplified manikin models were still retained, while the mesh elements required on the manikin surface were significantly reduced (over 50%) without noticeable computational errors (Li et al., 2015c).

![Figure 8.2: Original 3D scanned manikin (left); simplified manikin using mesh-decimating approach (right).](image)

The whole cabin domain including manikins and seats was discretised using unstructured mesh. To achieve accurate prediction of the airflow field in the vicinity of the manikins, grid size was locally refined in passengers’ micro-environment and 10 inflation layers with initial height of 1 mm were added on the manikin surfaces to capture the gradient change of velocity, temperature, etc. Four sets of mesh configurations were applied and tested prior to adding the contaminants, which required the total mesh elements of 6 million, 8 million, 11 million and 14 million, respectively. To achieve the mesh independence, all cases were firstly compared in terms of the mesh quality and grid convergence index (GCI) (Roache, 1994). The results indicated that after reaching 11 million of the mesh elements, further refinement of mesh did not produce significant improvement on the mesh quality and the GCI for finer grid (14 million) solution was less than 3%. The velocity predictions at different positions across the whole cabin domain were compared using the tested mesh configurations, as shown in Figure 8.3. Through the comparison, no considerable deviation on the velocity field was noticed after mesh elements were increased from 11 million to 14
million. Therefore, mesh configuration with 11 million mesh elements was adopted for the subsequent simulations.

![Figure 8.3: Mesh independence of velocity field.](image)

### 8.2.2 Boundary conditions and numerical setup

The ventilation rate at the inlets was set based on the American Society of Heating, Refrigerating and Air-Condition Engineers (ASHARE) aviation standard (ASHRAE, 2013). To mimic the worst case scenario, the minimum air supply of 9.4 L/s per person (Topp et al., 2002) was considered, which was in equivalent to the air mass flow rate of 0.04 kg/s at 20 °C inlet air temperature. Since passengers are the main heat source in the cabin, a convective heat load of 35.6 W was applied at each manikin, which was consistent with the existing literature (Topp et al., 2002) and our previous study (Yan et al., 2016). The front and back planes of the cabin section were assumed as translational periodicity, which added the periodic characteristics to
the airflow and particles leaving and re-entering through the set planes. Other solid walls, such as the floor, ceiling and seats were considered as adiabatic.

In terms of the disease transmission, contaminants were assumed to be released as sputum droplets through coughing. Coughing was considered as once-off release from passenger’s mouth. The evaporation process was no considered in this study because most sputum droplets would quickly evaporate (mostly within half second depending on the relative humidity) and become droplet nuclei with average diameter of 3.5 microns (Gupta et al., 2011; Chao et al., 2009). Since the airliner cabin is well-known as a low-humidity environment with relative humidity under 20% (Cui et al., 2014), the droplets would form to nuclei much quicker than other indoor environments. Therefore, a constant particle diameter of 3.5 microns was applied. The Lagrangian particle tracking model was employed to continuously trace the particle motions through the cabin domain, while particles were released by coughing to provide sufficient trajectories in the seven-row cabin section. The number of particles was tested prior to the case studies, as shown in Figure 8.4. 10,000 particles were found as the sufficient number to achieve consistent contaminants concentration. To consider the seating locations effects of the index patient, six representative cases were presented, in which every individual passenger (from A to F) sitting at the fourth row was successively assumed as the index patient in each case.

Figure 8.4: Sensitivity test of particle number along the longitudinal direction.
8.2.3 Mathematical models

The cabin airflow field was solved using the incompressible Navier-Stokes (N-S) equation, while the thermal buoyancy flow induced by the passengers’ body heat was considered through the Buossinesq approximation. For micro particle transport in the continuous air, the Lagrangian approach was employed to track the particle movement based on the equation of motion. Significant forces including the drag force \( \vec{F}_D \), the buoyance force \( \vec{F}_{Buoy} \), and the virtual mass force \( \vec{F}_{VM} \) were considered and expressed in Equations 8.2, 8.3 and 8.4.

\[
m_p \frac{d\vec{U}_p}{dt} = \vec{F}_D + \vec{F}_{Buoy} + \vec{F}_{VM} \tag{8.1}
\]

\[
\vec{F}_D = \frac{C_D \pi d_p^2}{2} \rho_a \left| \vec{U}_p - \vec{U}_a \right| \left( \vec{U}_p - \vec{U}_a \right) \tag{8.2}
\]

\[
\vec{F}_{Buoy} = \frac{\pi d_p^3}{6} (\rho_p - \rho_a) g \tag{8.3}
\]

\[
\vec{F}_{VM} = \frac{C_{VM} \pi d_p^3}{2} \rho_a \left( \frac{d\vec{U}_p}{dt} - \frac{d\vec{U}_a}{dt} \right) \tag{8.4}
\]

According to the report from existing literature (Liu et al., 2012), typical cabin environment has relatively low velocity and high turbulence, which means the main source that leads to the dispersion of the aerosol particles is the fluctuating component of the airflow. Thus, the turbulent dispersion of particle transport in the Lagrangian approach was modelled by adding an eddy fluctuating component onto the mean air velocity in conjunction with the entry of the particles. The local air velocity is redefined in Equation 8.5,

\[
\vec{U} = \bar{U} + U' \tag{8.5}
\]

where \( \bar{U} \) is the mean air velocity and \( U' \) is the fluctuating eddy velocity.

In each eddy, the fluctuating eddy velocity can be varied by the lifetime and the length of the eddy. The impact of the fluctuating eddy velocity on the particles is only valid when the following two conditions are met. Firstly, the interaction time between the entering particle and the eddy is shorter than the eddy lifetime. Secondly, the relative displacement of the particle to the eddy is less than the eddy length. If not, the fluctuating eddy velocity in this eddy is not considered and the particle is assumed to be directly entering into the next eddy with new lifetime, length and thereby the new fluctuating velocity (ANSYS, 2015).
\[ U' = \phi\left(\frac{2k}{3}\right)^{0.5} \quad (8.6) \]

\[ L_e = \frac{C_\mu^{3/4}k^{3/2}}{\varepsilon} \quad (8.7) \]

\[ t_e = \frac{L_e}{\left(\frac{2k}{3}\right)^{0.5}} \quad (8.8) \]

where \( \phi \) is a normal distributed random number which accounts the randomness of turbulence by a mean value. \( k \) and \( \varepsilon \) are the local turbulent kinetic energy and dissipation, respectively. \( C_\mu \) is the turbulent constant.

The N-S equations and particle tracking models were solved by CFX 16.2 (ANSYS, 2015). Steady computations of airflow and contaminants fields were conducted in conjunction with the RNG k- model for the air turbulence due to its successful application in modeling indoor airflow and pollutant transport (Isukapalli et al., 2013; Liu et al., 2012). Particles are assumed to be fully deposited when hitting the floors, seats and cabin walls, due to the factor that the materials applied on these boundaries in real cabins are high absorption materials (wool or nylon carpet, leather upholstery, fabric, etc.).

### 8.2.4 Risk assessment

Trajectory tracking of particles using the Lagrangian approach would provide very detailed and visualised transport history of the particles, which could give an idea of the possible deposition locations. However, it is insufficient to understand disease transmission only based on the transport characteristics of the particles. Concentration and distribution of particles are also essentially required to estimate the high risk regions. Since the Lagrangian approach only predicts the particle trajectories, the particle concentration was calculated based on the so-called particle source in cell (PSI-C) method (Zhang and Chen, 2007b) using Mathematica. The cabin domain containing the history of the particle trajectories was firstly discretised again using a number of control volumes (cells) and then the local particle concentration in a control cell was estimated based on the particle residence time, as expressed in Equation 8.9,

\[ C_j = \frac{M \sum_{i=1}^{n} dt(i, j)}{V_j} \quad (8.9) \]
where is the local particle concentration in the $j$th cell and is the volume of that cell. $M$ is the mass flow rate represented by a particle trajectory and $dt(i, j)$ is the residence time of the $i$th particle in the $j$th cell.

The Wells-Riley’s equation (Sze To and Chao, 2010) was utilised in conjunction with the CFD predictions to assess the infection risks of passengers.

$$P_I = 1 - \exp(-\frac{Iqpt}{Q})$$

where, $P_I$ is the probability of infection, $I$ is the number of infectors, which equals to 1 for single index patient case. $q$ is the quanta generation rate. For worst case scenario of infectious disease transmission (e.g. tuberculosis), $q = a$ unity infectivity term $\times$ number of quanta/unit time, in which passengers were assumed to be very vulnerable to pathogen. A unity infectivity term delineates that one quantum is equal to one infectious particle/pathogen (Sze To and Chao, 2010), which makes the model deterministic. $p$ and $t$ are passenger breathing rate and the exposure time interval, respectively.

8.3 Results and Discussion

8.3.1 Airflow Field and Model Validation

The experierntal data of airflow field by Li et al. (2015a) was firstly selected for model validation. In their study, a seven-row aircraft cabin mock-up was built inside a thermostatic chamber with seated thermal manikins to mimic the cabin environment of Boeing 737-200. The global airflow distribution and local velocity profiles were measured using large-scale 2D particle image velocimetry (PIV) technic. Their high resolution PIV measurements from both publication and supplementary materials provided very detailed data for validations. The velocity vectors measured at the fourth row of the cabin section (in front of the passengers) was selected and compared between the experimental measurements (Li et al., 2015a) and our numerical predictions, as illustrated in Figure 8.5. The velocity vectors predicted in this study yielded very similar airflow directions and distributions to the experimental results in most of the regions on this selected plane. It was noticable that the PIV measurements only cover the main region of this plane, while some of the spaces under the seats and between the roof racks were not included due to the limitation of experimental setups. Slight deviations were found at the corresponding edges of the PIV measurements, such as the airflow direction near the ground level. Despite some local
deviations, both experimental measurements and numerical predictions captured the same airflow pattern of the compared plane that two main circulations were formed after airflow injecting from the inlets and interacting at the aisle region.

Figure 8.5: Velocity vector comparisons between numerical predictions and experimental measurements (Li et al., 2015a) at row-4.

To quantitatively compare the airflow field, the predicted velocity profiles were further compared against the experimental data along 7 vertical lines, as shown in Figure 8.6. All vertical lines were extracted from the same plane given in Figure 8.5. The position and length of these lines were remained the same as those in Li et al. (2015a)’s experimental setup. In their study, arms of all manikin models were removed for the purpose of fitting experimental equipments, whilst the manikin models used in this study contained comprehensive body segments with full body features. The geometric difference of the applied manikin models could affect the predictions on the regions very close to the manikin body. Although deviations were noticable at some local sample points due to the manikin model difference, the overall numerical predictions were very close to the experimental data, especially at the aisle region (Line 4) where the affect of manikins were minimised, the predicted velocity profiles agreed very well with the experimental measurements.
Figure 8.6: Comparison of velocity profiles between numerical predictions and experimental measurements (Li et al., 2015a) at selected lines.

Since the airflow re-circulation exists at the entire cabin domain, it is important to assess whether the airflow patterns are regular at various cabin cross-sections (i.e. at different rows). The predicted airflow distributions were thereby compared along multiple cross-sections across the whole cabin domain. Four representative planes placed in front of passengers sitting at 1st row, 3rd row, 5th row and 7th row, respectively, were selected to demonstrate the results in Figure 8.7. The predicted results revealed that the airflow pattern is not entirely symmetrical along the horizontal direction (left to right), since the downward airflow was fluctuating unsteadily at the aisle regions. Similar asymmetric airflow field was also experimentally observed by Li et al. (2015a)’s in a cabin mock-up, which was believed to be induced by the random turbulent fluctuations of airflow in the cabin and the impact of the turbulent fluctuations was significantly enlarged with the increase length of cabin domain along the longitudinal direction. Therefore, to accurately investigate the contami-
nants transport which is mainly dominated by the airflow field, it is necessary to conduct investigations under a considerable large cabin domain.

![Diagrams of velocity distribution at different planes](image)

Figure 8.7: Velocity distribution at four selected planes in front of the passengers; a. Plane 1 (1st row), b. Plane 2 (3rd row), c. Plane 3 (5th row) and d. Plane 4 (7th row).

### 8.3.2 Particle Transport and Case Study

The particulate contaminants were assumed to be released through coughing by passengers to imitate the release of infectious diseases. In this study, a uniform droplet nuclei diameter of 3.5 \( \mu m \) was selected according to the study by Redrow et al. (2011). Since particles with diameter of 3.5 \( \mu m \) would be mainly dominated by the ventilated airflow inside the cabin and the local airflow profiles were found very different in front of different passengers, as can be noticed in Figure 8.6, the particle transport was expected to be very sensitive to the release location (the sitting location of the index patient). Therefore, in order to include the effect of index patient sitting locations, 42 computational cases were accomplished in this study, in which each passenger was considered as the sole index patient in one case study. Among all the cases, six representative cases were selected and presented to illustrate the particle transport and distribution characteristics. In these six cases, passengers sitting at row-4 were considered as the sole index patient successively, which allows the same longitude droplet travel distance range behind and in front of them, as described in Figure 8.8.
Figure 8.8: Case studies with different index patients.

Figure 8.9 illustrated the predicted particle transport after exhalation under different cases, in which particles were released through coughing by one of the passenger sitting at row 4 in each case. Through investigating the particle trajectories, it is noticeable that when the particles were released by passengers sitting at different locations, their transport characteristics were completely different. For passenger A and F, who were sitting just under the inlet air jet and above the outlet, the airflow velocity was relatively low at that region and was mainly dominated by passengers’ thermal plume. Since the thermal plume effect was significant in the cabin environment, in which passengers are sitting very close to each other, particles released by passenger A and F were quickly entrained up by the buoyancy driven thermal plume. Once particles were lifted high enough, it joined the main flow stream and then was completely dominated by the inject airflow. Therefore, the contaminants released by passenger A and F would travel much further and faster than the others due to the interaction with the ventilation jet. On the other hand, particulate contaminants released by passengers sitting closer to the aisle (B, C, D and E) travelled much slower and mainly suspended in front of the index patient and the neighbours. Since these passengers were sitting at the centre of the airflow re-circulation regions, the contaminants were mainly driven by the recirculating airflow. As a result, these particles would stay longer in passengers’ breathing zone. Contaminants seem to be locked inside the passengers’ breathing zone and hardly to be able to escape.
In order to quantify the difference of particle trajectories noticed in Figure 8.9, the travel distance of released particles by different passengers were carefully compared along the three coordinate directions (i.e. longitudinal, horizontal and vertical directions) and the results were plotted against the travel time, as illustrated in Figure 10. Every symbol plotted in Figure 8.10 represents the individual particle trajectory released by the index patient and different case studies were distinguished using different colour and shape of the symbols. The overall particle transport characteristics can be quickly compared using the fitted curves. It can be noticed that contaminants released by passenger A and F travelled much faster than the other cases at the first ten seconds along all directions. Although the travel direction was opposite between particles released by passenger A and F, the particle travel distances were very close between these two cases, which means the particle transport characteristics were similar when passengers sitting at sides of the cabin were coughing. It also can be noticed that when passengers sitting at the aisle seats (Passengers C and D) were coughing, particles experienced the shortest travel distance, especially along the horizontal direction (Figure 8.10b). This finding revealed that when aisle seats passengers were releasing harmful contaminants, the contaminants would be locked at themselves’ and their adjacent passengers’ breathing zones for very long time (more than 20 seconds in cases 3 and 4) under the particular ventilation scheme. This lock-up phenomenon
in passengers’ breathing zone could directly increase the exposure risk of passengers. Once harmful contaminants from other sources enter this lock-up region, the contaminants would not be able to leave the breathing zone easily due to the re-circulation and would eventually cause uncomfortableness or even serious health issues.

Figure 8.10: Particle travel distance along (a) longitudinal; (b) horizontal; (c) vertical directions.
8.3.3 Infection Risk Assessment

To assess the infection risk of passengers in cabin environment, the concentration of exhaled particles was firstly required to estimate the high-risk regions. The PSI-C method was referred to convert the particle trajectories into the concentration distribution using Mathematica. For each case, particle concentrations were firstly extracted along 30 cut planes (XZ-plane) at various heights inside the breathing zone and integrated into one normalised plane. All the normalised concentration distributions of particles yielded from six cases as aforementioned were demonstrated in Figure 8.11.

![Figure 8.11: Normalised concentration of exhaled particles (top view).](image)

It can be clearly observed that when different index patients were releasing contaminants, the particles were mostly concentrated at the same side of the cabin without travelling further across the aisle. The reason could be that the couple of large re-circulations (shown in Figure 8.7) split the airflow into two main domains (left and right), while passengers were mostly sitting at the centre of the re-circulations where the fresh air was not sufficient. The regions of higher risks can be easily and phenomenologically estimated through the concentration distribution in Figure 8.11.
For instance, when passengers B and E were releasing contaminants, normalised concentration distribution revealed that particles were highly concentrated around the passengers sitting nearby. However, if the particles after release were quickly brought away by the injected airflow (i.e. case 1) due to the release position, they would travel much quicker and further, while the concentration distribution would be less significant. Under this circumstance, it would be very challenging only relying on the concentration distribution to identify the high infection risk regions. Therefore, it is necessary to seek an alternative approach to quantifiably assess the infection risk for each individual passenger.

To achieve a quantifiable assessment of infection risks of each passenger, analysing the particle transport and distribution in passenger’s breathing zone is crucial. According to the Australia (2013), the breathing zone of each passenger was defined as a hemisphere of 300 mm radius extending in front of the face and the centre of the hemisphere was measured from the midpoint of the joining line between the ears. Detailed particle transport data (particle residence time, travel distance and etc.) in each passenger’s breathing zone was firstly extracted and then the infection risks were calculated and assessed based on the Wells-Riley equation (Equation 8.10). One index patient was included at various locations for each case. Other passengers were

![Figure 8.12: Infection risks in passengers’ breathing zones.](image)
assumed to be very vulnerable to pathogen, which set the quanta generate rate as a unity infectivity term multiply the number of quanta/unit time. Passengers’ breathing rate was carefully set based on the ASHRAE (2013), while the average particle residence time in the breathing zone was considered as the exposure time interval.

The assessed infection risks in each passenger’s breathing zone under different case were illustrated in Figure 8.12, in which the probability of infection was ranged from 0 to 1. The increase of the infection risks in passengers’ breathing zones could be directly reflected on the growth of the normalised figure, as well as on the change of colour from dark to light. The results shown in Figure 8.12 revealed that passengers sitting within 3-4 rows to the index patient would have very high chance to be infected in most cases. For case 2-5, extremely high infection risks were found in passengers’ breathing zones who were sitting adjacent to the index patient (same row and the next row). On the other hand, since the released particles were quickly suppressed and carried by the inject airflow in case 1, high infection risks were found a few rows behind the index patient. This finding indicated that passengers sitting far away from the index patient could also have high infection risks, although the particle concentration outside the breathing zone may not be significant. Once the passengers with high infection risks were identified, further evaluations can be conducted only in these passengers’ breathing zones, which would significantly accelerate the analysis and improve the efficiency. For example, in case 1, as passengers sitting at the left side of rows 6 and 7 were found with high infection risks, the particle transport

Figure 8.13: Particle trajectories in highly risked passengers’ breathing zones (case 1).
and distribution in their breathing zones can be further investigated in details, as
demonstrated in Figure 8.13. The results given in Figure 8.13 offered one approach
to investigate the particle transport and distribution in the breathing zone by using
light colour indicating high concentration of particles. More detailed investigations
inside passengers’ breathing zone in relation to the particle transport and distribution
is required in the future study, since the breathing zone is still a considerable large
space when passengers are sitting very close to each other under multi-occupied cabin
environment, although it is already significantly smaller comparing to the overall
domain of the airliner cabin.

8.4 Conclusion

This study employed a seven-row cabin model based on Boeing 737 to investigate
the airflow and particle transport characteristics in the cabin environments, followed
by the assessment of inflight infection risks. 3D characterised particle transport tra-
jectories were provided and discussed in conjunction with the comparison of particle
travel distances among six cases. The PSI-C method was used in this study to convert
particle trajectories into concentration, while the infection risks of passengers were
assessed using a quantifiable approach. The conclusions arising from this study are
the follows:

1. Particle travel distance was found to be very sensitive to the release locations
(i.e. released by passengers sitting at various locations), while the impact was
more significant along the longitudinal and horizontal directions. Particles re-
leased by passengers sitting at the window seats would travel much further
than the others. When passengers sitting closer to the aisle were coughing,
particles would suspend longer in the index patient’s breathing zone, as well as
the adjacent passengers.

2. A quantifiable approach based on the Wells-Riley equation was applied in this
study to assess the infection risks of inflight passengers. The approach is robust
as it focuses on the exposure risks in the breathing zone of every individual
passenger rather than the overall spaces. More importantly, this approach is
capable of providing a fast and direct assessment of the infection risks with
normalised results. When an index patient is found in the airliner cabin, the
probability of infection of the rest passengers will be quickly and accurately
assessed using this approach.
Also, an unsteady characteristic of the airflow pattern at the aisle region was noticed from this study, while a long cabin section was found to be necessary to capture this unsteady flow behaviour. This finding indicated that the size of the cabin section also plays an important role on when conducting simulations in cabin environment, whilst this factor was mostly compromised in existing study due to the high computational cost.

Overall, this study provided a systematic approach through not only combining the Wells-Riley equation in conjunction with the Lagrangian model in CFD, but also providing detailed and comprehensive analysis on the infection risks of every passenger. The ultimate intention of this study is to provide a systematic CFD approach in conjunction with the risk assessment model so that qualitative and quantifiable predictions and evaluations of infection risks would be achieved within a reasonable time of period (within a week). This would effectively help preventing further spread of the disease when index patient is determined.
Chapter 9

Conclusion

The holistic simulation of airliner cabin has been a challenging and long-existing task for many years due to the extreme complexity of the cabin environment induced by the multi-scale, multi-coupling and non-linear transport characteristics of contaminants in airliner cabins. Extremely high computational resources and cost are unavoidably required when conducting simulations under such complex cabin environments. As a compromise, some important affecting factors such as passengers thermal effect were eliminated, while passenger body features were missing through arbitrarily simplification approaches in the past. 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 novel and quantifiable manikin simplification algorithm was developed to reduce the computational costs without sacrificing accuracy.

- Comprehensive descriptions of inter-phase mechanisms were achieved in a cost-efficient way using E-E model to realise fast predictions of the PM concentration in airliner cabins.

- An unique technique to convert particle trajectories to concentration was developed and optimised based on the PSI-C method.

- A quantifiable approach to assess the infection risks in the airliner cabins was proposed.

- A systematic platform was developed to holistically assess the infection risks in airliner cabin environment.
9.1 Summary of the Contributions

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

9.1.1 Passengers’ Thermal Effects in Occupied Airliner Cabin

In Chapter 4, a section of airliner cabin containing 3 seats and 3 passengers to investigate the thermal effects of passenger body heat on the airflow field and the transport characteristics of exhaled droplets. The simulations were conducted with isothermal and thermal conditions and the numerical results were validated using experimental data and compared against each other. Conclusions arising from this study are as follows:

1. The thermal buoyancy flow driven by the passenger body heat has a significant effect on the overall and local airflow fields in the cabin section. The thermal plume effect was maximised in some local regions, e.g. in front of passengers, between two passenger shoulders and above passenger heads under typical cabin environment. The intensity of thermal plume may vary among different passengers, depending on the sitting locations of passengers and cabin geometry.

2. The transport and distribution characteristics of the droplets exhaled by the passengers were highly sensitive to the location of release. When droplets were released by the passenger close to the window (Passenger A), they may have longer residence time in other passengers breathing zones.

9.1.2 Validated Mathematical Models for PM Transport

The Eulerian-Eulerian two-phase flow model was employed in Chapter 5 to model the transport and concentration distribution of particulate matters. Computations were conducted with both transient and steady states, and both isothermal and thermal conditions. The model was validated using the experimental data available in the literature and compared against the existing two-phase flow models for PM transport, in the aspects of accuracy and computational cost. Conclusions arising from this study are as follows:

1. The Eulerian-Eulerian model has a comparable accuracy with the Lagrangian model and performs better than the drift-flux model, this is especially true when the particle concentration is relatively high and the particle size is large (e.g. PM10) when significant particle settling or deposition could happen.
2. When the PM concentration is preferred, the Eulerian-Eulerian model has its unique advantage over the Lagrangian model as it gives a direct prediction to the PM concentration, thus does not need any additional post-processing procedures. This not only largely reduces the computational cost, but also eliminates the uncertainties that might be caused by the additional post-processing procedures.

9.1.3 Manikin Simplification Effects for CFD Simulations

Computational thermal manikins (CTMs)/Computer simulated persons (CSPs) were simplified using three different approaches (i.e. surface smoothing approach, the skeleton-based approach (Ruzic and Bikic, 2014) and surface-area-based approach (Miyanaga et al., 2001)) in Chapter 6. The effect of thermal manikin diversity by simplifications on predicting the thermal airflow fields were studied under quiescent condition, displacement and mixed ventilations. Based on the outcomes, the conclusions rising from this study are as follows:

1. The predicted airflow in the thermally affected region was highly sensitive to the approach of the manikin simplifications, although manikin simplifications did not have any detectable effects on the airflow prediction in the bulk region. The significance of manikin model variety will be enlarged on top of the manikin head due to the effect of buoyancy driven thermal plume by body heat, while the locations of the maximum velocity were very sensitive to the applied manikin models.

2. The geometrical diversity of CTM caused by simplifications had higher impact on the thermal airflow field under the mixed ventilation than that with displacement ventilation based on the studied cases. This outcome may vary if the vent sizes and positions are changed or the inlet air velocity is different.

3. The CTM simplified through surface smoothing approach achieved a very close prediction as compared to the baseline case with an error of less than 5%, whereas the predictive errors associated with the skeleton-based and the surface-area-based manikin simplifying approaches were 14.8% and 18.1%, respectively. Using the surface smoothing approach, the required number of the mesh elements on the CTM surface required to achieve mesh-independency could be
reduced by 75%, which would certainly contribute to an improved computational efficiency while maintaining a reasonable predictive accuracy. The surface smoothing based approach was recommended for the future work.

9.1.4 A Quantifiable Manikin Simplification Approach for Large Cabin Environment

In Chapter 7, a quantifiable manikin simplification approach using mesh-decimating algorithm was developed based on the outcomes from Chapter 6. Effects of passenger model simplifications on both airflow and contaminants fields were evaluated under airliner cabins. Gaseous contaminants were assumed to be released from the manikin skin surfaces. The distribution of the contaminants was studied and compared among various CTMs. Conclusions arising from this study are as follows:

1. The proposed manikin simplification approach using mesh-decimating algorithm provides a quantifiable, controllable and cost-efficient way to simplify 3D-scanned CSP models, which allows a flexible simplification control based on the given conditions. In addition, the simplified CSPs allow coarser mesh for CFD computations, without compromising the numerical stability and accuracy, which further improves the computational efficiency.

2. Although the effect of manikin simplification on the prediction of airflow field was mainly detected in the occupants thermally-affected regions, it could cause significant inaccuracy in the contaminant concentration field in the whole domain. However, through controlling the iteration number or the simplification index of manikin simplification, the accuracy of using a simplified manikin model is still comparable to that of using a 3D-scanned manikin model. The computations demonstrated that even for a densely occupied indoor space such an airliner cabin, an iteration number of 20 or a SI less than $3.5 \times 10^{-4}$ is still safe to maintain the accuracy. Therefore, this approach is recommended as an efficient and promising approach that can not only guarantee the accuracy of the numerical results, but effectively reduce the computational cost.

3. The contaminants released at lower height regions can be potentially brought upward by the buoyancy driven thermal plume. As a result, considerable amount of contaminants were detected in the occupants breathing zone, while the highest contaminant concentration occurred atop the manikin head. The
effect of the thermal plume would be maximised under multi-occupants environment and thereby cause much higher concentration of contaminants inside the occupants breathing zones and eventually lead to serious health hazards. Proper design of HVAC setups is strongly required and the personalised airflow device may be necessary to help avoiding contaminants suspending in the breathing zone.

9.1.5 Infection Risk Assessment in Fully Occupied Airliner Cabin

Chapter 8 employed a seven-row cabin model based on Boeing 737 to investigate the airflow and particle transport characteristics in the cabin environments, followed by the assessment of inflight infection risks. 3D characterised particle transport trajectories were provided and discussed in conjunction with the comparison of particle travel distances among six cases. The PSI-C method was used in this study to convert particle trajectories into concentration, while the infection risks of passengers were assessed using a quantifiable approach. The conclusions arising from this study are the follows:

1. Particle travel distance was found to be very sensitive to the release locations (i.e. released by passengers sitting at various locations), while the impact was more significant along the longitudinal and horizontal directions. Particles released by passengers sitting at the window seats would travel much further than the others. When passengers sitting closer to the aisle were coughing, particles would suspend longer in the index patients breathing zone, as well as the adjacent passengers.

2. A quantifiable approach based on the Wells-Riley equation was applied in this study to assess the infection risks of inflight passengers. The approach is robust as it focuses on the exposure risks in the breathing zone of every individual passenger rather than the overall spaces. More importantly, this approach is capable of providing a fast and direct assessment of the infection risks with normalised results. When an index patient is found in the airliner cabin, the probability of infection of the rest passengers will be quickly and accurately assessed using this approach.

3. Also, an unsteady characteristic of the airflow pattern at the aisle region was noticed from this study, while a long cabin section was found to be necessary
to capture this unsteady flow behaviour. This finding indicated that the size of the cabin section also plays an important role on when conducting simulations in cabin environment, whilst this factor was mostly compromised in existing study due to the high computational cost.

This study provided a systematic approach through not only combining the Wells-Riley equation in conjunction with the Lagrangian model in CFD, but also providing detailed and comprehensive analysis on the infection risks of every passenger. The ultimate intention of this study is to provide a systematic CFD approach in conjunction with the risk assessment model so that qualitative and quantifiable predictions and evaluations of infection risks would be achieved within a reasonable time of period (within a week). This would effectively help preventing further spread of the disease when index patient is determined.

Overall, by integrating mechanistic multi-phase flow models, novel manikin simplification approaches and 3D dynamic characterisation of contaminant transport, a systematic and cost-efficient platform was thereby developed for comprehensive assessments of air quality and particulate contaminant transport in airliner cabins. The outcomes of this research laid an important and solid foundation for air quality optimisation and health risks assessment in other densely occupied spaces (high-speed rail, metro and etc.).
Bibliography


ASHRAE (2009). Air quality within commercial aircraft. ASHRAE.


