Numerical Simulation and Indoor Airflow and Heat Transfer Study for Thermal Comfort

Presented by Kana Horikiri

Submitted in Partial Fulfilment of the Requirements For the Degree Doctor of Philosophy

Supervised by Professor Yufeng Yao

Faculty of Science, Engineering and Computing Kingston University London

March 2015



IMAGING SERVICES NORTH

Boston Spa, Wetherby West Yorkshire, LS23 7BQ www.bl.uk

PAGINATED BLANK PAGES ARE SCANNED AS FOUND IN ORIGINAL THESIS

NO INFORMATION IS MISSING

I

Acknowledgment

The work described in this thesis was carried out by the author in Faculty of Science, Engineering and Computing, Kingston University, under the supervision of Professor Yufeng Yao.

I would like to thank all academic and technician staff of the School, and others, who helped and encouraged me in the course of this work, and in particular:

Professor Yufeng Yao and Dr Jun Yao for giving me this opportunity to carry out this Ph.D program study and for their enduring support, advice and motivation during the whole course of research.

Mitsubishi Electric R&D Europe MERCE-UK for the research scholarship.

Faculty of Science, Engineering and Computing for partial financial support.

Finally, I would like to especially thank Arash for providing endless support, joy and motivation in my life.

Kana Horikiri

III

Contents

Acknowl	ledgment	II
Contents		IV
Abstract		ΊΠ
List of Fi	igures	. X
List of Ta	ablesX	IV
Chapter 1	1 Introduction	1
1.1.	Facts of energy consumption in dwellings	1
1.1.1	1. Energy consumption in dwellings	. 1
1.1.2	2. Heating systems in dwellings	3
1.1.3	3. Costs in energy efficient house	. 5
1.2.	Airflow and heat transfer	. 6
1.3.	Flow unsteadiness in indoor environment	. 8
1.4.	Conjugate heat transfer	. 9
1.5.	Thermal comfort	10
1.6.	Major contributions to the field	12
1.7.	Scope of this thesis	13
Chapter 2	2 Mathematical Model of Fluid Flow, Heat Transfer and Thermal Comfort	14
2.1.	Governing equations	15
2.2.	Heat transfer in solid domain	16
2.3.	Radiation model	17
2.4.	Turbulence model	18
2.5.	CFD Solver	20
2.6.	Computational grid generation and convergence study	21
2.7.	Fast Fourier Transformation (FFT)	21
2.8.	Indoor thermal comfort	22
2.9.	Theory and physical parameters and dimensionless variables	24
2.9.1	Buoyancy-driven flow theory	24
2.9.2	2. 2-D conjugate heat transfer model study	25
2.9.3	3. 3-D conjugate heat transfer model study	26
2.9.4	4. 3-D thermal comfort study	27

2.10. Sur	nmary	28
Chapter 3 H	Benchmark Cases	29
3.1. Tw	o-dimensional flow and heat transfer benchmark cases	29
3.1.1.	Forced convection in a 2-D empty room (Case 3.1.1)	30
3.1.2.	Natural convection in a 2-D tall cavity (Case 3.1.2)	32
3.1.3.	Mixed convection in a 2-D empty room (Case 3.1.3)	36
3.1.4.	Summary of 2-D benchmark case studies	39
3.2. Th	ee-dimensional flow and heat transfer benchmark cases	39
3.2.1.	Forced convection in a 3-D empty room (Case 3.2.1)	42
3.2.2.	Forced convection in a 3-D room with a unheated box (Case 3.2.2)	45
3.2.3.	Mixed convection in a 3-D room with heated box (Case 3.2.3)	46
3.2.4.	Summary of 3-D benchmark case studies	49
3.3. Flo	w unsteadiness in a 3-D benchmark case	49
3.3.1.	Oscillation signal	49
3.3.2.	Power spectral density (PSD) analysis	50
3.3.3.	Summary of flow unsteadiness	61
3.4. Sur	nmary of airflow and heat transfer study	62
Chapter 4 C	Conjugate Heat Transfer (CHT) on Indoor Thermal Comfort	64
4.1. Cor	njugate heat transfer in two-dimensional closed model	64
4.1.1.	General description	64
4.1.2.	Validation	66
4.1.3.	Summary of two-dimensional CHT study	69
4.2. Cor	ijugate heat transfer in three-dimensional model	. 70
4.2.1.	General description	. 70
4.2.1.1	Non-CHT model room	. 70
4.2.1.2	2 CHT model room with finite-thickness wall	. 72
4.2.2.	Validation	. 74
4.3. Para	ametric study in a 3-D CHT model room	. 78
4.3.1.	Effects of heat source and window glazing sizes (Case 4.3.1)	. 80
4.3.2.	Effect of wall thickness (Case 4.3.2)	. 82
4.3.3.	Effect of wall thermal conductivity (Case 4.3.3)	. 83
4.4. Sum	ımary	. 84
Chapter 5 T	hermal Comfort and Heat Transfer in Furnished and Occupied Room	. 86
5.1. Hea	t transfer and thermal condition in furnished and occupied room	. 86

5.1.	.1. General description	86
5.1.	2. Validation	92
5.2.	Case study	93
5.2.	.1. Effect of furniture arrangement (Case 5.1.1)	93
5.2.	2. Effect of heat transfer from occupant (Case 5.1.2)	95
5.2.	3. Effect of heat transfer from heat-generating furniture (Case 5.1.3)	101
5.3.	Numerical optimisation of thermal comfort	102
5.4.	Summary	106
Chapter	6 Conclusion and Future Work	109
6.1.	Summary	109
6.2.	Conclusion	109
6.3.	Contributions	112
6.4.	Future Work	113
Referenc	es	114

VII

Abstract

An investigation of indoor thermal environment has been carried out by computational fluid dynamics approach. The study focuses on the thermal comfort evaluation, particularly the flow and heat transfer effects due to conjugate natural convection, furniture arrangement and occupant number, and flow oscillations. Key physical features of thermo-fluid such as velocity and temperature distributions, thermal sensation maps, and oscillation frequency and its energy are quantified, analysed and compared.

The benchmark case study of airflow and heat transfer showed that ANSYS Fluent RNG $k - \epsilon$ turbulence model with temperature boundary condition on the heated boundary calculated the best results, compared with available data. It also showed that air velocity increased along the boundary walls and especially hot wall which led flow direction upwards. At the centre of the flow circulation, air momentum is very weak (e.g. almost zero velocity magnitude). The increase of complex features (e.g. a box with/without heat) in the domain would lead to flow separations causing recirculations above the box and in the rear space of the domain and swirls in the front space presenting three-dimensional flow, and a thermal plume, compared with a two-dimensional clockwise flow in an empty room. The flow recirculations and thermal buoyancy enhanced velocity magnitude and turbulence level in the domain. In fact, the highest frequency was obtained in the room with an unheated box, followed by the room with a heated box. The formation of thermal plume from the heated box stabilised the flow in the upper part and the sides of the heated box on a spanwise plane. The frequency of velocity oscillation was consistent with temperature at the location although the energy of the fluctuation is much higher in temperature. Moreover the dominant frequency depended on the orientation of the flow circulation, for example a high energy at a lower frequency on a spanwise plane while a low energy at a higher frequency on a streamwise plane. In an empty room, it was found that there is no direct relation in an empty room (case 3.2.1) between velocity and turbulent flow in power spectral density and frequency, and each of time-history velocity oscillations is independent and random. At the midheight of the domain, the energy of the velocity fluctuation is relatively weak.

The results from the study of conjugate natural convection heat transfer in a ventilated room with localised heat source and window glazing showed that the size of heat source and window glazing, the wall thickness and wall material property are important factors to temperature change and heat loss. For example, 30 % of wall thickness reduction caused 35 % more of heat loss through the wall and 9 % of comfort temperature.

From the study of furniture arrangement and occupant number in a 3-D model room with localised heat source and window glazing, it was found that the presence of furniture induced flow recirculation and higher velocity around furniture and the presence of thermal occupant formed thermal plume in the fluid domain, increasing volume-averaged temperature by maximum 15 %, compared with that of unoccupied and empty model room. Increase in the number of occupants and thermal furniture helped increase air temperature by 6.5 %, compared with that of single occupant and the averaged PPD (Predicted Percentage of Dissatisfied) value around the occupants by maximum 5.4 % for one occupant and 11.5 % for two occupants, respectively. The location of occupant was very sensitive to flow stream path, e.g. the PPD distribution was symmetrical in the spanwise position but became asymmetrical in streamwise position. Further investigation of thermal comfort level using Fanger's indices due to ventilation rate and thermal load led that desirable indoor environment might be achieved with higher ventilation flow rate $(U_{inlet} > 0.7 m/s)$ rather than reducing heat generation from the heating sources for more occupants introduced to the room.

The results in the thesis summarise some of the important reservations with regard to the CFD capability and reliability for indoor thermal environment and present data would be useful for the built environment thermal engineers in design and optimisation of domestic rooms.

List of Figures

Figure 3.1. Sketch of forced convection airflow in a 2-D model room
Figure 3.2. Comparison of x-component velocity at (a) $x = H$ and (b) $x = 2H$ on mesh
topology, compared with experiment (Restivo, 1979)
Figure 3.3. Sketch of the natural convection flow in a 2-D room
Figure 3.4. Comparison of vertical velocity (left) and temperature (right) profiles at
Y = 0.1, 0.5 and 0.9 in different turbulence models compared with experiment (Betts and
Bokhari, 2000)
Figure 3.5. Sketch of the mixed convection flow in a 2-D room
Figure 3.6. Velocity vector (a) and temperature (b) distributions of 2-D mixed convection
flow
Figure 3.7. Comparison of velocity (a) and temperature (b) profiles at $X = 0.5$ with
experiment (Blay et al. 1992)
Figure 3.8. Sketch of a 3-D model room with dimensions and inlet/outlet slot (a) and ten
measuring positions along two central lines in horizontal x-z plane (b)
Figure 3.9. Comparison of normalized velocity magnitude (U) vectors/contours between
experiment (top) and CFD prediction (bottom) at a middle plane $(x, y, z = 0.5L)$ 43
Figure 3.10. Comparison of normalized turbulence kinetic energy K contours between
experiment (top) and CFD prediction (bottom) at a middle plane of $x, y, z = 0.5L$ 43
Figure 3.11. Comparison of normalised velocity and turbulence kinetic energy
distributions at positions 2, 5 and 8 (case 3.2.1) with experiment (Wang and Chen, 2009).
Figure 3.12. Comparison of normalised velocity and turbulence kinetic energy
distributions at positions 2, 5 and 8 (case 3.2.2) with experiment (Wang and Chen, 2009).
Figure 3.13. Comparison of predicted temperature contours at 20,000 iterations (bottom)
on a streamwise (a,c) and spanwise (b,d) mid-planes with experiment (Wang and Chen,
2009) (top) (case 3.2.3)
Figure 3.14. Comparison of normalised velocity, turbulence kinetic energy and
temperature predictions at positions 1, 6 and 8 (case 3.2.3) with experiment (Wang and
Chen, 2009)

Figure 3.15. Time serials of velocity magnitude and temperature at $Y = 0.5$ at P10 for
three cases
Figure 3.16. Energy spectra of time-history velocity magnitude fluctuations at monitoring
locations P1, P3, P5 for case 3.2.1
Figure 3.17. Energy spectra of time-history velocity magnitude fluctuations at monitoring
locations <i>P6</i> , <i>P</i> 10 for case 3.2.1
Figure 3.18. Energy spectra of time-history velocity magnitude fluctuations at monitoring
locations <i>P</i> 1, <i>P</i> 3, <i>P</i> 5 for case 3.2.2
Figure 3.19. Energy spectra of time-history velocity magnitude fluctuations at monitoring
locations <i>P</i> 6, <i>P</i> 10 for case 3.2.2
Figure 3.20. Energy spectra of time-history velocity magnitude fluctuations at monitoring
locations P1, P3, P5 for case 3.2.3
Figure 3.21. Energy spectra of time-history velocity magnitude fluctuations at monitoring
locations <i>P6</i> , <i>P10</i> for case 3.2.3
Figure 3.22. Energy spectra of time-history temperature fluctuations at monitoring
locations P1, P3, P5 for case 3.2.3
Figure 3.23. Energy spectra of time-history temperature fluctuations at monitoring
locations P6, P10 for case 3.2.3
Figure 4.1. Schematic view of a 2-D model room
Figure 4.2. Streamlines (Ψ) and isotherms (θ) at $\tau = 500$ for (a, c) $Gr = 1.6 \times 107$ and
(b, d) $Gr = 2.4 \times 107$
Figure 4.3. Comparisons of stream function and non-dimensional temperature profiles at
two Gr numbers, two vertical locations of $Y = 0.35$ (a, c) and $Y = 0.8$ (b, d) and
$\tau = 500$
Figure 4.4. Schematic view of 3-D CHT configuration
Figure 4.5. Comparison of comfort temperature profile at monitoring lines with
FloVENT76
Figure 4.6. Comparison of temperature variation at $x = 0 m$ at (a) $z = 0 m$ and (b)
z = 0.9 m.
Figure 4.7. Distributions of (a) isotherms ($T \circ C$) in the solid wall and (b) temperature
distributions at three vertical heights compared to theoretical estimation at a streamwise
mid-plane by the CHT model

Figure 4.8. Variations of heat transfer through the radiator as heat source (two bottom
lines) and the corresponding comfort temperature (two top lines) for three different sizes
of radiator (<i>hBH</i>) and window glazing (<i>h2H</i>)
Figure 4.9. Variations of heat transfer (solid lines) at solid wall surfaces and the
corresponding comfort temperature (dashed line) at different wall thickness as dL83
Figure 4.10. Variations of heat transfer (solid lines) at solid wall surfaces and the
corresponding comfort temperature (dashed line) at different wall thermal conductivity
ratios kairkwall
Figure 5.1. Schematic views of four 3-D configurations of furniture with monitoring four
lines $(P1 - P4)$: (a) layout S0, (b) layout S1, (c) layout S2, (d) layout S388
Figure 5.2. Schematic view of seated occupant
Figure 5.3. Detail of seated occupant's segments (a) side-view, and occupied zone with
eight measuring points (b) top-view
Figure 5.4. Comparison of comfort temperature and PPD profiles at four monitoring
locations $P1 - P4$ between present study and published FloVENT results (Myhren and
Holmberg, 2008), (Myhren and Holmberg, 2009)93
Figure 5.5. Velocity magnitude iso-surface of $0.1 ms$ coloured with x-velocity (u)
contours for four room layouts: (a) S0, (b) S1, (c) S2, (d) S394
Figure 5.6. Comparison of comfort temperature profile at four monitoring locations
P1 - P4 from four room layouts $S0 - S3$ with that of FloVENT for an empty model
room layout S0 (Myhren amd Holmberg, 2009)95
Figure 5.7. Air temperature distributions at a mid-streamwise plane ($z = 0 m$) for three
layouts with sofa: (a) S1, (b) S2, (c) S397
Figure 5.8. Velocity contours at the mid-width of occupant body for three layouts with
sofa: at $x = 4.25 m$ (a) S1H1 and (c) S3H1 and at $z = 0.85 m$ (b) S2H2and (d) S3H2.
Figure 5.9. PPD predictions for seated occupants H1 (in red lines) and H2 (in blue lines)
in different layouts with sofa (S1 and S2 in solid lines, while S3 in dashed line) at four
vertical locations (a) $Y = 0.1 m$, (b) $Y = 0.5 m$, (c) $Y = 1.1 m$ and (d) $Y = 1.3 m$,
respectively
Figure 5.10. Comparison of averaged PPD values around occupants (H1, H2) and
volume-average air temperature for three layouts with sofa (S1, S2, S3) with TV "on-
mode" and "off-mode"

Figure 5.11. PPD predictions for seated occupants H1 (in red line) and H2 (in blue line)
in three layouts with sofa (S1 and S2 (a, c, e, g) and S3 (b, d, f, h)) for case 5.3.1 (in
dash-dotted-dotted lines), case 5.3.2 (in solid lines) and case 5.3.3 (in dashed lines) at
four vertical locations (a, b) $Y = 0.1 m$, (c, d) $Y = 0.5 m$, (e, f) $Y = 1.1 m$ and (g, h)
Y = 1.3 m respectively 105
r = 10 m, respectively.
Figure 5.12. Comparison of average PMV values around occupants (H1, H2) and
Figure 5.12. Comparison of average PMV values around occupants ($H1, H2$) and volume-average air temperature from three layouts with sofa ($S1, S2, S3$) for cases 5.3.1,

List of Tables

Table 1.1. Average percentage of heat transfer from radiator (Peach, 1972)	4
Table 2.1. Relationship between PMV and thermal sensation	24
Table 3.1. Summary of 2-D benchmark cases	29
Table 3.2. Boundary conditions of 2D forced convection study	31
Table 3.3. Boundary conditions of 2D natural convection study	33
Table 3.4. Boundary conditions of 2D mixed convection study	37
Table 3.5. Summary of benchmark cases in three-dimensional	39
Table 3.6. Location of monitoring points	41
Table 4.1. Specification of a 3-D model room	71
Table 4.2. Boundary conditions of non-CHT model	72
Table 4.3. Comparison of fluid temperature and heat transfer from the radiator	75
Table 4.4. Comparison of surface temperature and heat transfer of the wall	75
Table 4.5. Parametric case studies	80
Table 5.1. Case study with heat generation source	87
Table 5.2. Specifications of furniture and human body	90
Table 5.3. Boundary conditions	91
Table 5.4. Parametric case studies	103

xv

Chapter 1

Introduction

1.1. Facts of energy consumption in dwellings

1.1.1. Energy consumption in dwellings

The transformation of energy market in transports, industries, appliances and buildings is challenging but compulsory in order to tackle global warming as well as other pressing environmental issues. In building sector alone, energy consumption is responsible for almost 40 % of total energy consumption in Europe (European Commission: EU energy policy) (Boermans, T., Petersdorff, C., 2007). In the UK, the largest contribution to energy consumption in indoor environment is space heating, risen by 24 % over last 30 years since 1970 (Department of Trade and Industry, 2002). On the other hand, between 2000 and 2012 energy consumption by the domestic sector fell by 7 % (Department of Energy and Climate Change, 2014). The energy consumption per household, which is more closely related to the size of dwelling than the numbers of population and household, has decreased about 6 % by year 2000 (Department of Trade and Industry, 2002) since the average floor-area size of household is declined in the UK. However the number of households has been increasing as well as population while the population per household declined since 1970. This reflects that more people live alone in smaller houses. In fact, the rate of energy consumption per person has increased by 18 % at year 2000 since 1970. Two people living in two houses consumes the same amount of energy as two people living in the same household according to the study from national statistics by Department of Trade and Industry (Department of Trade and Industry, 2002).

With the increased interests in energy-conscious and sustainable eco-buildings development for better indoor environment and less energy consumption, the common approaches in cutting energy consumption is to reduce heat loss by introducing insulations in cavity walls and lofts, using double glazing of window, and improving or replacing aged heating systems. For example, insulations are applied for 48 % of the UK dwellings by 2008 in cavity wall for which the insulation lies inside the layers of wall at

construction process and only 2 % of solid walled dwellings in the pre-1920 houses which add internal or external extra layers along the wall (Department for Communities and Local Government, 2010). The reason of a small number of popularity of solid wall dwellings is due to cost and extra wall thickness. On the other hand, loft insulation has been also introduced to more dwelling, increasing about 13 % by 2008 from the 25 % level in 2003 (Department of Trade and Industry, 2002). The large change for better energy efficiency in dwelling was seen in the use of double glazing windows, due to the fact that heat transfer through window takes approximately 33 % of heat loss in cold winter by air infiltration at edges of the windows and cold glass surface. After new building regulations introduced in 2006, the number of dwelling with fully double glazing windows rapidly increased from 30 % in 1996 to 71 % in 2008.

Insulation and efficiency of heating systems are two most important factors of heating energy usage and are related to house age. However older dwellings with insulation tend to be difficult to have the same level of thermal performance as the new houses. Change in heating systems also cannot guarantee to improve efficiency in the old dwellings due to original wall structure. The only feasible way of mostly increasing the level of energy efficiency and reducing the level of CO_2 emissions per house is to update those houses dating after 1976 with modern insulation and heating systems and to introduce more updated homes (Palmer and Cooper, 2012). The newer domestic buildings considered from 1990 relatively improve 'average' energy efficiency based on each dwelling's energy cost per square metres by about 20 points from 47 points in dwellings built pre-1945 with no updated insulations. Although the trend of improving energy efficiency is clearly seen in new-built dwellings, only 14 % of the dwellings were developed after 1984 in the UK (Department for Communities and Local Government, 2010).

The types of dwelling bring a large impact to energy consumption. Detached houses including bungalows generally require more energy than flats due to large surface-areato-volume-ratio (e.g. in year 2008 the average floor area of detached house in the UK was $143 m^2$ while it was 70 m^2 for flats, which is equivalent to 1.7 bedrooms (Nationwide, 2008)), equivalent to 10 points higher in 'average' energy efficiency in flats than the average energy efficiency in all dwelling types (Department for Communities and Local Government, 2010). Using Energy Efficiency Rating (EER) that is the information of

how well insulated the dwelling is and of which heating efficiency in energy performance certificates set by the government, the flats in this case are equivalent to 'D' band houses, whereas 'E' band is for detached houses and bungalows.

Energy bills are one of the most household spending factors. Any energy users pay more bills than ever even if energy cost savings are concerned. As mentioned earlier, the rate of energy consumptions in dwellings was at the same level as before or even slightly decreased. The price of electricity that is the main source of space heating has increased by 20 % since 1970 (Palmer and Cooper, 2012). The rate of average household consumption of standard electricity from 2008 to 2012 was -9.26 % in kWh, representing energy consumption savings. But the costs of actual standard electricity consumption from 2010 to 2012 increased 5.94 % in pounds, which reflects average fixed costs of £ 60 per household in 2012 (Barrington, 2013). The energy bills are directly affected by the price of fuels. The trend of price increase in electricity therefore is linked to the increasing price of oil, i.e. the input fuel.

1.1.2. Heating systems in dwellings

The common system for space heating in the UK is central heating with radiator panels, consist of boiler and radiator in most distributing heat from pipes containing hot water boiled in the boiler to the radiator in the dwelling, accounting for approximately 90 % of all households, equivalent to 21.7 million homes in Great Britain by 2000 (Department of Trade and Industry, 2002). Central heating systems can distribute almost uniform temperature in whole building. Some 71 % of the central heating systems installed in houses were supplied by gas in 2000. This development stretched average room temperature from 13 °C to 18 °C. The number of central heating with condensing boiler increased to 17 % by 2008 since 2004 (Department for Communities and Local Government, 2010). This is due to change in building regulations that it became mandatory for all replacement. Condensing boiler uses a large heat exchanger to obtain more heat than other types of boiler such as standard, back and combination, and hence is the efficient boiler type. Hence the declines of trend in standard and back boilers have seen over last 15 years and combination boilers also started to be replaced by condensing boilers.

Location, size and pipe layout are important for thermal transfer performance. Material and surface conditions are also influential factors. Radiators are normally placed underneath windows so that thermal plume can block cold air from ventilation slot between the window and the ceiling. In order to maximise heat output from a radiator, it is necessary to decrease the space between the radiator and the floor, to increase the space between the radiator and the adjacent wall, to introduce fins within the radiator, or even to place the reflector on the adjacent wall, etc.

The materials of radiator are commonly made in cast iron, steel and aluminium. Aluminium panel is good for start-up heating due to its good thermal conductivity and light weight. On the other hand, cast iron is good for continuous heating but does not have a good quick response. It contains a large amount of water and hence requires more operating cost. It is very heavy-weight panel. Steel panels are used in convectors, panel radiators and towel radiators in bathroom. It is more economic due to less water content in the panels and more number of available panels according to owner's requirements. Table 1.1 shows heat transfer in convection and radiation modes from different types of radiator in the isothermal surface enclosure. Although convection is the main heat transfer method from any radiators listed below, radiative heat transfer helps provide more comfort environment in the space. Radiant heat brings warm air at relatively uniform temperature through the space from the floor where it is mostly needed. On the other hand, convective heat transfer leads more temperature stratification as cold air stays at lower levels and warm air at higher levels.

	Average proportion of total heat transfer (%)	
Types of radiator	Convection	Radiation
2-column cast iron radiator	70	30
3-column cast iron radiator	78	22
Convector/Radiator (stainless steel)	85 — 90	10 — 15
Single panel (stainless steel)	50	50
Double panel (stainless steel)	70	30

Table 1.1. Average percentage of heat transfer from radiator (Peach, 1972)

1.1.3. Costs in energy efficient house

It is true that the cost of new energy efficient building is generally higher because of many factors such as planning and guideline for construction sector, education and training for designers, and quality assurance including valuable tools bring up costs as well as time. The cost difference in construction between standard new house and low energy house can be seen only 2 - 6% in Germany, Austria, Sweden and Switzerland while 3 - 10 % in the UK (European Comission, 2009). It is 8 % difference (around 15,000 Euro) in construction costs between a low energy house and more energy-efficient house such as passive house for Germany in 2009 (European Comission, 2009). However it will show significant energy saving over the lifetime of low energy houses compared to standard new buildings since a low energy house use 15 - 25% of the energy required to run a conventional house. In order to introduce more energy-efficient houses, some countries in Europe support financial incentives for development towards low energy buildings (Jensen et al, 2009). It will not be easy for existing houses to reduce the whole costs in building energy performance as well as energy consumption and CO_2 emissions. This is because the amount of jobs and plans that an existing house needs for energy efficiency is not equal to those in better energy performance houses and low energy houses and the number of existing houses is far more than that of energy efficient houses.

Impacts on energy performance can be seen most in replacing traditional boiler with condensing boiler and in installing insulations in loft and cavity wall recommended in the Energy Performance Certificate (EPC) methodology. This can be explained by the fact that boilers account for 55 % of annual energy bill. Hence high efficiency condensing boiler could save up to \pounds 310 per year (Energy saving trust, 2013).

The development of insulation reduced the energy consumption by 59 % in 2000, compared to that in 1970 and will result in 5 -10 % of energy saving by decreasing one degree in heating system (European Commission, 2006). But professional loft installation in the existing house costs about £ 300 - £ 500 and saves up to £ 180 in energy bills per year in the UK. It is £ 140 energy saving from cavity wall insulation. This is equivalent to between 560 kg to 700 kg of CO_2 emission savings per year. On the other hand, as mentioned earlier, insulations over solid wall are comparable, costs over £ 5,000

in total installation but save up to £ 490 in energy bills per year which is equivalent to 1.9 tonnes in reducing CO_2 emissions per year (Energy saving trust, 2013).

The guideline and requirements for energy performance in buildings vary from country to country in Europe. It is hence difficult to compare the absolute values describing energy performances among different countries. However the targets towards improvement of building energy performance and minimising CO_2 emissions from buildings are unified and confirmed in the terms of low energy house, passive houses and zero energy houses.

In encountering the largest challenge to improve energy performance in building sector, one of the main issues is flow, heat transfer and energy conservation in the room space. Temperature control by mechanism such as convection, conduction and radiation, designing of thermal plumes by location of heating system and others, and understanding of surface conditions have the potential to reduce CO_2 emissions from space heating, to cut energy bills in the most expensive energy price, to lead ideal thermal conditions for occupiers.

1.2. Airflow and heat transfer

Indoor environment design often depends on details of air velocity and temperature distributions, relative humidity levels, contaminant concentrations, and turbulence levels, among many other factors. Most indoor airflows are very complicated and often driven by a combination of pressure, buoyancy and viscous forces. There are three classified convectional modes; i.e. forced convection such as the injection of cooling airflow by using an air-conditioning unit in the summer, natural convection such as the buoyancy effect of heated air flow by radiators in the winter, and/or mixed convection such as the air movement due to different types of heating systems (e.g. air-conditioning) in operation in the summer or winter. In order to accurately predict the airflow movement and thermal field properties, one of two methods is generally used, namely experimental measurement and computational simulation. Most experiments adopt a full-scale test chamber to produce an artificial indoor environment to isolate the internal room space from the external effects. While this method permits the control air and thermal boundary

conditions, the cost of construction is high and the test could take very long time to complete. With advances in computer power and numerical methods, an alternative approach using computational fluid dynamics (CFD) has been increasingly used to model indoor airflow. CFD can provide much more detailed field information such as velocity, temperature and turbulence kinetic energy distributions than experiments at lower cost and in a shorter time.

In the past decades, considerable progresses have been made in this field, as reviewed by Chen et al. (Chen et al, 2010). There are varieties of benchmark cases that have been studied by various researchers. For example, Zuo and Chen (Zuo and Chen, 2009) studied two-dimensional (2-D) forced, natural and mixed convections by using a novel fast fluid dynamic method, and for the same 2-D models, Zhang et al. (Zhang et al, 2007) further studied turbulence model effects. For industrial applications, the Reynolds-averaged Navier-Stokes (RANS) equations with an eddy-viscosity based turbulence model is still a dominant methodology, due to its reasonable accuracy and fairly short turn-around computing time of both CPU and wall-clock time, as discussed by various researchers (Wang and Chen, 2009), (Jiang and Chen, 2003), (Zhang et al, 2007). One area of research interests is the study of indoor air movement since this is related to health care and human comfort level in a room. For example, Shih et al. (Shih et al, 2007) performed a case study of an isolated hospital room environment in protecting patients from the spread of air contaminated with bacteria. The common features of these studies are the quantification of suitable key factors, such as mesh resolution, its quality and topology, turbulence model and near wall treatment, before reliable predictions can be obtained by design engineer.

For a general 3-D problem such as the evaluation of thermal comfort level in an indoor environment, the physics behind the fluid dynamics and the heat transfer would be complex, because of the nonlinearity and time-dependence of the problem. For example, in the cold winter season, a ventilation system is required to improve the indoor thermal conditions of the room, as well as to improve the air quality by air circulation. Thus, the interaction between the 'cold' airflow from the ventilation system intake and the 'warm' airflow from the heating systems would have significant effect on heat transfer characteristics that will ultimately impact on the thermal comfort level (Myhren and Holmberg, 2009), the flow structures (Raji et al, 2008) and air quality (Rim and Novoselac, 2010).

1.3. Flow unsteadiness in indoor environment

In indoor ventilated spaces, airflow contains many complex movements, especially around obstacles and wall corners. The airflow therefore is usually turbulent and the air velocity is fluctuating against time. In indoor environment, however, there are very limited number of reports available related to flow unsteadiness in the past. So far, one of the achievements in the field is to have correlated between velocity fluctuation and discomfort to occupant. The discomfort has shown a maximum frequency in the range of 0.3 - 0.5 Hz (Fanger et al, 1977). Other researchers claimed from their experiments that the turbulent energy spectrum has a dominant frequency in the interval 0 - 0.2 Hz in the ventilated spaces with/without thermal load (Thorshauge, 1982) and up to 10 Hz in the ventilated spaces (Hanzawa et al, 1985). Other researchers studied natural convection in a square cavity (Tian amd Karayiannis, 2000), observing different frequencies between velocity and temperature fluctuations and variation of frequency in the flow direction along the isothermal walls. A characteristic frequency in a vertical free convection was numerically found at 0.35 Hz (Gebhart and Mahajan, 1975). Turbulent flow fluctuation in a core of a cavity was reported (King, 1989) due to strong asymmetrical flow caused by heat loss. For oscillation study in mixed convection, a correlation between Reynolds number (*Re*) and Grashof number (*Gr*) was found in $700 \le Re \le 1000$ and $10^3 \le Gr \le 1000$ 10⁶ (Beya and Lili, 2007). In ventilated spaces for occupants, the relationships between mean velocity and turbulence intensity was found that mean velocity increases as turbulence intensity and domain height decrease (Thorshauge, 1982) (Hanzawa et al, 1985) (Kovanen et al, 1989). The possible causes of flow instability could be thermal conductivity of boundaries (Henkes and Hoogendoorn, 1990) and strength of buoyancy (Sinha et al, 2000), and limited in boundary layers along the solid walls (Tian and Karayiannis, 2000).

The traditional approach of building a dedicated physical test room for field measurement of key parameters such as velocity and temperature would be very expensive and time consuming, and also the measured data are often limited although it is vital for providing accurate reference data for indoor environment engineers. Computational Fluid Dynamics (CFD) technique has demonstrated much strong capability in capturing the detailed information of real-time 3-D complex fluid and heat transfer characteristics at both the system and component levels in a domain. It is also able to carry out post-processing of a large amount of information, which saves cost, time and space in engineering design process. The major problem on flow oscillation in indoor environment is that due to vast amount of data and time based on onsite measurements the detailed information such as oscillation energy level and fluctuation change across the domain.

1.4. Conjugate heat transfer

Analysis of conjugate natural and forced convective flows and heat transfer performance of built environment has been an interesting research subject. It is because of its technical applications in design and layout of indoor thermal devices, heat storage systems and indoor thermal environment comfort assessment, among many other reasons. However, the coupling of fluid flow and heat transfer would be complex and challenging even for a single natural convection model room. This is due to the nonlinearity of the physical problem itself and also the interactions between the closely related flow field and temperature field. Recently, there have been growing demands from building industry and heating thermal device design sector in analysing and quantifying accurate information of flow and thermal characteristics of a typical indoor environment for human beings. One of many key steps towards the ultimate goal of eco- or smart-building design is to have a thorough understanding of flow and heat transfer phenomena in relation to thermal heat sources and layout, material properties and boundary conditions of room walls and surfaces. This is because they will have a major influence on indoor thermal comfort level including air quality and heating/cooling loads.

Past researches were primarily focused on the heat transfer and thermal effects in a relatively simple two-dimensional (2-D) model room such as an enclosed domain without a heat sink. Their investigations were performed on flow patterns, fluid temperature distributions and the relation of Nusselt number (Nu) and Rayleigh number (Ra) with respect to heated walls or heating systems (Kaminski and Prakash, 1986), (Ntibarufata et

al, 1994), (Ben Nasr et al, 2006), (Muftuoglu and Bilgen, 2008), (Kuznetsov and Sheremet, 2011). The relation of Nu and Ra also describes heat transfer rate and ventilation rate in a 2-D tall ventilation cavity (Gan, 2011). A common conclusion from these studies was that at a Ra number, $Ra \leq 10^7$, the heat transfer performance in terms of Nusselt number is proportional to Rayleigh number and also dependent on the thermal conductivity ratio of the fluid and the solid. Similar findings were reported in studies of different 2-D conjugate natural convection configurations (Bilgen, 2009), (Saeid, 2007), in which the addition of a vertical heated plate would significantly reduce the heat transfer rate, from about 40 % for thin walls to about 12 % for thick walls (Bilgen, 2009). For large Grashof number $Gr > 10^5$, the temperature inside the finite-thickness wall was broadly of two-dimensional distribution and the non-uniform distributed temperature on the solid-fluid interfaces would cause asymmetric flow patterns (Kaminski and Prakash, 1986). The distribution of heat flux was also affected by surface radiation emissivity, wall conductivity ratio, and wall thickness (Nouanegue et al, 2008). In the conjugate mixed convection study, it was revealed that the locations of vertical heat source and horizontal ventilation opening slot would have major influences on the strength and pattern of flow circulation and the level of heat transfer (Koca, 2008). Despite most of the twodimensional conjugate heat transfer studies have shown basic fluid flow and heat transfer characteristics using stream lines and heat lines (Deng and Tang, 2002), (Kaluri et al, 2010), there are limited studies on modelling more general and complicated flow and heat transfer features in a three-dimensional (3-D) configuration. Furthermore, there are not many studies on analysing the relationships between indoor thermal condition and conjugate conduction and convection heat transfer performance.

1.5. Thermal comfort

The transformation of energy market in transports, industries, appliances and buildings is challenging but compulsory in order to tackle global warming as well as other pressing environmental issues. In building sector alone, which is responsible for almost 40% of total energy consumption (European Commision: EU energy policy), the interests in energy-conscious and sustainable eco-buildings development have been increasingly grown to have better indoor environment and less energy consumption. As a result, there have been numerous indoor thermal environmental studies, for example indoor environment of transportation (Alahmer et al, 2012), (Li et al, 2014), public spaces/buildings (Pourshaghaghy and Omidvari, 2012), (Alfano et al, 2013), (Barbhuiya and Barbhuiya, 2013), workspaces/offices (Hens, 2009), whole building environment (Bos and Love, 2013), (Park et al, 2014), specific enclosed space (Jang et al, 2007), among many others. The factors that influence indoor thermal comfort level are ventilation systems for ventilation effectiveness (Krajcik et al, 2012), (Olesen et al, 1980), (Deuble and de Dear, 2012), (Pereira et al, 2009), air distribution (Bos and Love, 2013), wall thickness and thermal insulations (Kumar and Suman, 2013), glazing systems (Buratti et al, 2013), (Cappelletti et al, 2014), fluid temperature of heat sources (Tye-Gingras and Gosselin, 2012), and radiant temperature (Atmaca et al, 2007).

One common feature of these studies is about the thermal comfort evaluation and assessment. In general, thermal comfort can be described by available models such as Standard Effective Temperature (SET) (Gagge et al, 1971), comfort temperature (Myhren and Holmberg, 2009) and Predicted Mean Vote (PMV) - Predicted Percentage of Dissatisfied (PPD) (Fanger, 1972). Of which, the Fanger's indices (i.e. combined PMV and PPD) have been widely adopted as the so-called ISO 7730 standard (International Organization for Standardization, 2007) due to the well-correlated human factors and environmental parameters, and also the adaptability in many types of buildings, except for some very special ventilation types (Deuble and de Dear, 2012). The method predicts thermal sensation and thermal discomfort quantitatively, based on environmental parameters (e.g. air temperature, mean radiant temperature, air velocity and air humidity) and thermal balance of a human being (e.g. physical activity and clothing) obtained by either field measurements or numerical calculations. This model has also been used to develop other thermal comfort sub-models (MacArther, 1986), (Scheatzle, 1991), (Federspiel and Asada, 1994), (Fanger and Toftum, 2002), (Homod et al, 2012), design and optimise building spaces under specific weather/climate conditions (Wei et al, 2010), (Andreasi et al, 2010), and study thermo-fluid characteristics in space/room without objective and human occupant (Myhren and Holmberg, 2008), (Tye-Gingras and Gosselin, 2012) or with objective and human occupant (Lan et al, 2014), respectively.

In addition to aforementioned factors, indoor thermal comfort is also sensitive to other physical parameters in case of room environment with and without occupant. For example, surface temperatures of human body and room walls can cause the increase of radiation temperature (Atmaca et al, 2007). The orientation and surface treatment of window glazing (Buratti et al, 2013) and occupant behaviour (Selens et al, 2011) also have important effects on energy performance and thermal balance of thermal comfort level. The careful monitor and control of fluid inlet temperature of a heating radiator panel can maximise the indoor thermal comfort as well as minimise the energy consumption (Tye-Gingras and Gosselin, 2012). The PMV calculation, after considering all these environmental factors, has exhibited decreased levels of sensitivity, from very significant to mean radiant temperature, down to less significant to air temperature and velocity and finally to insignificant to air humidity (Alfano et al, 2011), respectively. The changes in outdoor climate and season will also influence indoor thermal comfort (Frontczak and Wargocki, 2011).

1.6. Major contributions to the field

As discussed in the previous sections, the information of indoor thermal transfer in a space that occupant being relax (e.g. a domestic living room) is still limited and uncertain. This thesis aims to provide much more detailed information of convection and conduction heat transfer in complex and realistic situations and its thermal comfort level in order to increase reliability and accuracy in the gaps between detailed physics and quantitative findings. The findings would be useful for the built environment thermal engineers in design and optimisation of domestic rooms with a heat source.

The major contributions to indoor environment are as follows.

- Providing detailed numerical information of heat transfer in complex room configuration
- Finding correlation between indoor thermal comfort temperature and heat loss through wall under the UK building regulations
- Predicting Fanger's indices around occupants in thermally complex domestic room
- Providing correlation between thermal comfort level and the number of occupant

- Finding the sensitivity of Fanger's index (e.g. PPD) to stream path in furnished room
- Finding correlation between flow oscillation and room configuration, and velocity fluctuation and temperature fluctuation
- 1.7. Scope of this thesis

This research of numerical investigation of indoor thermal heat transfer for improvement of thermal comfort will be carried out with the following key factors.

- Characteristics of flow and heat transfer in 3-D model room with a heat source (Chapters 3)
- Flow oscillations and its energy (Chapter 3)
- Wall conditions and heat source size (Chapter 4)
- Thermal/non-thermal furniture arrangement and occupant (Chapter 5)
- Ventilation rate and thermal load of heat source (Chapter 5)

The detailed objectives of this thesis are:

- To develop CFD model to simulate the characteristics of flow and heat transfer in indoor environment (Section 3.1 and Section 3.2)
- To investigate flow oscillation and its energy in forced and mixed convection heat transfer (Section 3.3)
- To predict the effect of finite-thickness wall conditions and location and size of heat source and window glazing on convective and conductive heat transfer (Chapter 4)
- To investigate heat transfer in living space and condition (e.g. furniture and occupant) (Section 5.3)
- To predict and optimize the thermal comfort level using Fanger's indices by improving the ventilation rate and heat energy (Section 5.4)

Chapter 2

Mathematical Model of Fluid Flow, Heat Transfer and Thermal Comfort

The traditional approach of building a dedicated physical test room or even a complete test house for onsite real-time measurement of key physical parameters such as temperature of the fluid (air) and the solid wall is still valid and vital for providing accurate reference data for building thermal design engineers. However, this approach would be technically challenging, very expensive and time consuming (Deuble and de Dear, 2012), and also the measured data are often limited, so that they cannot be easily applied for some specific configurations (Dascalaki et al, 1995). Simple correlation based calculation often used by building industry provides another option but it is generally not accurate enough for a comprehensive evaluation (Cengel and Ghajar, 2011).

With the advancement of numerical method and computational power, modern Computational Fluid Dynamics (CFD) techniques provide an alternative way of obtaining 3-D time-dependent flow and thermal parameters at both system and component levels (Wang and Chen, 2009), (Jiang et al, 2003), (Chen et al, 2010), (Jin et al, 2013). Furthermore, CFD can produce much detailed information to optimise an existing or a future thermal design and to perform thermal comfort assessment of an indoor environment (Myhren and Holmberg, 2009), (Chu et al, 1976), (Lu et al, 1997). The fast growing computer technology and architecture such as multi-core CPU and GPU make CFD even more feasible to carry out vast number of parametric studies as precursor numerical exercises and thus to integrate modelling work with practical engineering design and analysis process for cost saving, durability and reduced time scale from product design to market, for which it is almost impossible with physical tests and measurements due to extremely long preparation and construction time, and high operation and labour costs. For these reasons, numerical predictions have increasingly become an important element integrated in any engineering design and analysis for cost saving, durability and reduced time scale from product design to market.

A commercial CFD code, ANSYS FLUENT was used to calculate the heat transfer and its data was used to calculate comfort temperature (Myhren and Holmberg, 2009) defined in user-defined function in Fluent and thermal comfort indices by an in-house Fortran code which is developed based on Fanger's thermal environment model (Fanger, 1972) for assessment indoor thermal environment. Heat transfer considered in this study is convection through air and conduction though one-layer solid domain. The effect of radiative heat was included to account for both scattering and exchange of radiation between fluid and finite-thickness solid domains. Due to complex flow feature, some turbulence models were studied for validation and verification.

2.1. Governing equations

The fluid flow and heat transfer is governed by a set of conservation equations (mass, momentum and energy). The momentum Navier-Stokes equation is used for laminar flow in 2-D conjugate heat transfer problem and Reynolds-averaged Navier-Stokes (RANS) equation is adopted for turbulent flow in other 2-D and 3-D problems. These equations are expressed in a general Cartesian form as follows:

Mass conservation equation

~

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \, \vec{v}) = 0 \tag{1}$$

Momentum conservation equation

$$\frac{\partial}{\partial t}(\rho\vec{v}) + \nabla \cdot (\rho \,\vec{v} \,\vec{v}) = -\nabla \,p + \nabla \cdot (\bar{\tau}) + \rho \,\vec{g} + \vec{F}$$
⁽²⁾

Energy transport equation

$$\frac{\partial}{\partial t}(\rho E) + \nabla \cdot \left[\vec{v}(\rho E + p)\right] = \nabla \cdot \left(k_{eff} \nabla T - \sum_{j} h_{j}^{*} \vec{j}_{j} + \left(\vec{\bar{\tau}}_{eff} \cdot \vec{v}\right)\right) + S_{h}$$
(3)

where t is time, ρ is density (kg/m^3) , ∇ is partial differentiation operator, \vec{v} is velocity vector, p is pressure (Pa), $\rho \vec{g}$ is gravitational body force vector and \vec{F} is other external

body force vector, E is total energy (W), k_{eff} is effective heat conductivity (W/mK), T is temperature, h^* is sensible enthalpy, $\int_{T_{ref}}^{T} c_p \, dT$ (J), c_p is specific heat at constant pressure j/kgK, \vec{J}_j is diffusion flux of species j, $\bar{\tau}_{eff}$ is effective stress tensor, S_h is an additional energy source due to chemical reaction or radiation. The right-hand side of the energy equation represent energy transfer due to conduction, species diffusion, and viscous dissipation, respectively. The term of $\bar{\tau}$ is written as

$$\bar{\tau} = \mu \left[\left(\nabla \, \vec{v} + \nabla \, \vec{v}^{\,\mathrm{T}^*} \right) - \frac{2}{3} \, \nabla \cdot \vec{v} \, l^* \right] + S_{turbulent} \tag{4}$$

where μ is viscosity (kg/sm), T^* is matrix transpose, I^* is unit tensor, $S_{turbulent}$ is Reynolds stress term for turbulent flow $(S_{turbulent} = \nabla \cdot (-\rho \overline{\nu' \nu'})$, where $-\rho \overline{\nu' \nu'}$ is Reynolds stress tensor). The first term of the right-hand side is the effect of volume dilation.

2.2. Heat transfer in solid domain

The energy transport equation in the solid region used by ANSYS Fluent is expressed as

$$\frac{\partial}{\partial t}(\rho h^*) + \nabla \left(\vec{v} \rho h^*\right) = \nabla (k \nabla T) + S_h \tag{5}$$

where k is thermal conductivity (W/mK).

The second term on the left-hand side of Eq. (5) represents convective energy transfer due to rotational or translational motion of the solid. The velocity field \vec{v} is computed from the motion specified for the solid zone. The terms on the right-hand side are the heat flux due to conduction and volumetric heat sources within the solid, respectively.

The temperature distribution within the solid region is governed by 1-D heat conduction equation as

$$\nabla^2 T = 0 \tag{6}$$

At the interface between fluid region and solid region in the conjugate heat transfer model, the conductive heat transfer throughout the solid is coupled with the convective heat transfer in the fluid by

$$(\nabla\theta)_{fluid} = \frac{k_{wall}}{k} (\nabla\theta)_{wall}$$
(7)

where θ is dimensionless temperature and k_{wall} is wall thermal conductivity.

2.3. Radiation model

Due to the existence of a heating source in the computational domain, radiation heat needs to be included in the energy equation (3) via the source term S_h . In present study, a Discrete Ordinates (DO) model (Chandrasekhar, 1960) is adopted due to radiation temperature support and relatively wide range of applications within moderate computational cost and it has been already implemented in ANSYS Fluent software by incorporating the enthalpy balance to account for radiative heat transfer from a given heating source to adjacent medium (e.g. fluid) via a finite number of trajectories, each associated with a vector direction \vec{s} defined in the global Cartesian coordinate system. The solution of DO model is similar to that of fluid flow and energy transport equations and the resultant heat flux will be coupled with the energy equation through source term S_h in Eq. (3). In the DO model, the radiative heat transfer equation for an absorbing, emitting, and scattering medium at a position \vec{r} in the direction \vec{s} is given by

$$\nabla(I(\vec{r},\vec{s})\vec{s}) + (a + \sigma_s)I(\vec{r},\vec{s}) = an^2 \frac{\sigma T^4}{\pi} + \frac{\sigma_s}{4\pi} \int_0^{4\pi} I(\vec{r},\vec{s}') \Phi(\vec{s},\vec{s}') d\Omega'$$
(8)

where *I* is radiation intensity (W/sr) and is dependent on the position \vec{r} and the direction \vec{s} , *a* is absorption coefficient, σ_s is scattering coefficient, *n* is refractive index, σ is Stefan-Boltzmann constant $(5.669 \times 10^{-8} W/m^2 K^4)$, Φ is phase function and Ω' is solid angle (sr). The Discrete Ordinates (DO) radiation model is applied with various angular discretisation and sub-iteration parameters to control angles in discretising each octant of the angular space and volume overhang on each surface respectively, so that radiative conditions can be applied to individual faces and fluid within the domains.

2.4. Turbulence model

There are a number of turbulence models available for characterised flow feature. Large-Eddy Simulation (LES) was developed for large-scale of the flow (Smagorinski, 1963) and fully resolves only large-scale eddies which is anisotropic while the small-scale eddies, isotropic, are filtered from the turbulence flow. Hence this approach reduces a great amount of computer capacity which is overloaded by fine grid and small time steps, compared with Direct Numerical Simulation (DNS) solving the Navier-Stokes equations for flows with moderate Reynolds number (Ozgokmen et al, 2009). Reynolds average numerical simulation (RANS) calculates the statistic characteristics of the turbulent motion by averaging the flow equations over a time scale. The average turbulent flow calculation is feasible with a coarse mesh for steady state flow, resulting in less expensive to solve. Among the RANS turbulence models, they are the standard (Launder and Spalding, 1972), RNG (Yakhot and Orszag, 1986), realizable $k - \varepsilon$ (Shih et al, 1995) and shear stress transport $k - \omega$ (Menter, 1994).

Turbulent flow in indoor environment is generally a mixed small and large eddies in low Reynolds numbers, for which flows the standard $k - \varepsilon$ model overestimates the turbulent diffusivity. The turbulence model in indoor airflow simulations was studied by Chen and Zuo (Chen, 1995), (Zuo and Chen, 2009), concluding that the RNG $k - \varepsilon$ model showed good performance in accuracy, numerical stability and reasonable short computing time for low Reynolds number flow in two-dimensional cases, even with a very larger grid aspect ratio in the simulation.

In the benchmark cases, two-equation turbulence models, the re-normalized group (RNG) $k - \varepsilon$ model and the shear stress transport (SST) model are adopted to take account of the Reynolds stress and turbulent heat flux for momentum and energy equations, considering grid resolution, computing time and flexibility. The two-equation re-normalized group RNG $k - \varepsilon$ turbulent model is similar to the standard $k - \varepsilon$ model, the turbulence kinetic energy (k) and its dissipation rate (ε), and the model transport equation for k is derived from the exact equation while the model transport equation for ε is obtained using physical reasoning and bears little resemblance to its mathematically exact counterpart. The RNG is derived by using renormalization group methods (Yakhot

and Orszag, 1986). The small scale eddies are eliminated via the RNG method and introduced into modified Navier-Stokes equations which comprise a modified turbulent viscosity, a modified force and a modified non-liner coupling (Fluent, 1993). The Shear Stress Transport (SST) $k - \omega$ model (Menter, 1994) is an model based on model transport equations for the turblence kinetic energy (k) and the specific dissipation rate (ω), which can be the ratio of ε to k, and found to be a good approach for the present modelling, especially near walls with a relative coarse mesh.

The RNG $k - \varepsilon$ turbulence model (Yakhot and Orszag, 1986) is described below.

$$\frac{\partial}{\partial t}(\rho k) + \nabla(\rho k \vec{v}) = \nabla(\alpha_k \,\mu_{eff} \,\nabla k) + G_k + G_b - \rho \varepsilon - Y_M + S_k \tag{9}$$

$$\frac{\partial}{\partial t}(\rho\varepsilon) + \nabla(\rho\varepsilon\vec{v}) = \nabla(\alpha_{\varepsilon}\,\mu_{eff}\,\nabla\varepsilon) + C_{1\varepsilon}\,\frac{\varepsilon}{k}\,(G_k + C_{3\varepsilon}\,G_b) - C_{2\varepsilon}\rho\frac{\varepsilon^2}{k} - R_{\varepsilon} + S_{\varepsilon} \quad (10)$$

where k is turbulence kinetic energy, μ_{eff} is effective viscosity, ε is turbulence dissipation rate, G_k is turbulence kinetic energy generation with respect to mean velocity gradients, G_b is turbulence kinetic energy generation with respect to buoyancy, Y_M is dilatation dissipation, $C_{1\varepsilon} = 1.42$, $C_{2\varepsilon} = 1.68$ and $C_{3\varepsilon}$ is constant, $\alpha_k, \alpha_{\varepsilon}$ are inversed 'effective' Prandtl numbers for k and ε , and S_k, S_{ε} are source terms. The R_{ε} term accounts for the effect of rapid strain and streamline curvature change, $R_{\varepsilon} = {C_{\mu}\rho\eta^3(1-\eta/\eta_0)/1+\beta\eta^3}(\varepsilon^2/k)$ where $\eta \equiv Sk/\varepsilon, \eta_0 = 4.38, \beta = 0.012$.

The RNG model is responsive to the effect of rapid train and streamline curvature in $\eta > \eta_0$ thus the R term is a negative contribution in Eq.(10). Turbulence viscosity in the RNG equation varies with the scale of Reynolds number, for example, turbulence viscosity $d(\rho^2 k/\sqrt{\epsilon\mu}) = 1.72(\hat{v}/\sqrt{\hat{v}^3 - 1 + C_v})d\hat{v}$ is calculated with $\hat{v} = \mu_{eff}/\mu$ and $C_v \approx 100$ for low Reynolds numbers while turbulence viscosity is $\mu_t = \rho C_{\mu} k^2/\epsilon$ with $C_{\mu} = 0.0845$.
The Transport equations for the SST $k - \omega$ model are (Menter, 1994):

$$\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_i}(\rho k u_i) = \frac{\partial}{\partial x_j} \left(\mu + \frac{\mu_t}{\sigma_k} \frac{\partial k}{\partial x_j} \right) + \tilde{G}_k - Y_k \tag{11}$$

$$\frac{\partial}{\partial t}(\rho\omega) + \frac{\partial}{\partial x_i}(\rho\omega u_i) = \frac{\partial}{\partial x_j}\left(\mu + \frac{\mu_t}{\sigma_\omega}\frac{\partial\omega}{\partial x_j}\right) + G_\omega - Y_\omega + D_\omega$$
(12)

where turbulent viscosity is defined as

$$\mu_t = \frac{\rho k}{\omega} \frac{1}{\max\left[\frac{1}{\alpha^*}, \frac{\partial F_2}{\alpha_1 \omega}\right]}$$
(13)

In these equations, \tilde{G}_k represents the generation of turbulence kinetic energy due to mean velocity gradients, G_ω represents the generation of ω , $(\alpha/\nu_t)/G_k$, α represents a coefficient, Y_k and Y_ω represent the dissipation of k and ω due to turbulence defined by $Y_k = \rho \beta^* k \omega$ ad $Y_\omega = \rho \beta \omega^2$, respectively, D_ω represents the cross-diffusion term representing blend of the standard $k - \omega$ model and the standard $k - \varepsilon$ model, α^* represents a coefficient, ϑ represents the strain rate magnitude, σ_k and σ_ω represent the turbulent Plandtl numbers, $1/[(F_1/\sigma_{k \text{ or } \omega,1}) + \{(1-F_1)/\sigma_{k \text{ or } \omega,2}\}]$, respectively, F_1 and F_2 represent blending functions.

In a two-dimensional conjugate heat transfer (CHT) case study, laminar viscous model is used due to low Reynolds number and for three-dimensional case studies including CHT case; turbulent viscous flow model is adopted with two-equation renormalized group (RNG) $k - \varepsilon$ turbulence model.

2.5. CFD Solver

The aforementioned equations are solved numerically by finite volume method on structured grid. An iterative solution method, SIMPLE algorithm (Patankar and Spalding, 1972), is employed to solve the linearity of the momentum equation, the velocity-pressure coupling and the coupling between the flow and the energy equations. For pressure Poisson equation, the solution applies weighted body-force under the assumption

that the gradient of the difference between the pressure and body forces is constant, especially in buoyancy calculations. Other equations such as momentum, energy and radiation are solved using the second-order numerical scheme. For all the cases, numerical accuracy of double precisions is defined and the residual target is defined as 10^{-12} to achieve a high level of accuracy. The appropriate under-relaxation factors were imposed to avoid instability in the solution. In the benchmark cases, a hybrid finite element/volume method (ANSYS CFX) and unstructured mesh were applied for comparison of results. Like most commercial CFD package, ANSYS CFX converts the governing partial differential equations into a system of discrete algebraic equations by discretizing the computational domain and uses a couple solver solving the fluid and pressure as a single system.

2.6. Computational grid generation and convergence study

The grid designates the cells or elements on which the flow is solved and is a discrete representation of the geometry of the problem and has cells grouped into boundary zones where boundary conditions are applied. On the other hand, It has a significant impact on rate of convergence (or even lack of convergence) and solution accuracy and CPU time required for the simulation. In this study, software Gambit was used to generate mesh. The baseline of grid numbers/points and their distributions were close to that of references (e.g Zuo and Chen (Zuo and Chen, 2009) for 2-D forced and natural convections), and the grid was refined or coarsened by factor of 0.5, 1.5, 2, 3, etc with finer meshes clustered in inlet and outlet, and near wall regions. The grid refinement study was performed until a consistent prediction among grid resolutions was obtained, and grid independent solutions are shown in the results. The number of grid points is described in each problem.

2.7. Fast Fourier Transformation (FFT)

The investigation of flow unsteadiness in a space is evaluated using the Fast Fourier Transform (FFT) technique, based on predicted time-history data (e.g. velocity and temperature) from ANSYS Fluent at monitor locations. The FFT algorithm is developed to reduce the computation time of Discrete Fourier Transform (DFT) for N points from

 N^2 to Nlog 2(N) (Cooley and Tukey, 1965). The time-history data is transformed to frequency of flow oscillations. The basic idea is to break up a transform (i.e. DFT) of length N into two transforms of length N/2, shown below. For this study Matlab programmes is used to calculate frequency and power spectral density of flow oscillations. The Discrete Fourier Transform (DFT) is described as

$$X_N(k) = \sum_{n=0}^{N-1} x(n) \omega_N^{kn}$$
(14)

And

$$x(n) = \left(\frac{1}{N}\right) \sum_{k=0}^{N-1} X_N(k) \omega_N^{-kn}$$
⁽¹⁵⁾

Splitting into two

$$X_N(k) = \sum_{r=0}^{N/2-1} x(2r) \omega_N^{2rk} + \sum_{r=0}^{N/2-1} x(2r+1) \omega_N^{(2r+1)k}$$
(16)

Then the Fast Fourier Transform (FFT) is described as

$$X_N(k) = \sum_{r=0}^{N/2-1} x(2r) \omega_{N/2}^{rk} + \omega_N^k \sum_{r=0}^{N/2-1} x(2r+1) \omega_{N/2}^{rk}$$
(17)

where

$$\omega_N^2 = e^{2j(-2\pi)/N} = e^{j(-2\pi)/(N/2)} = \omega_{N/2}$$
(18)

2.8. Indoor thermal comfort

The thermal comfort indices are evaluated by using Fanger's comfort equations (Fanger, 1972), i.e. predicted mean vote (PMV) and predicted percentage of dissatisfied (PPD), representing the thermal balance of a whole human body. The parameter PMV is an index representing the mean value of the voters of a large group of people in the same environment on a seven-point thermal sensation scale, i.e. -3 < PMV < +3, see Table 2.1. The parameter PPD is also an index representing the percentage of thermally

dissatisfied persons among a large group of people. For thermal comfort requirement, the recommended PMV and PPD values are in a range of -0.5 < PMV < +0.5 and PPD < 10 %, respectively. Following the work of Fanger (Fanger, 1972), PMV and PPD values can be calculated by equations below. The equations are solved in FORTRAN programme with obtained results in CFD (e.g. air velocity, air temperature and radiation temperature). Radiation temperature with the DO radiation model is computed over a finite number of discrete solid angles associated with a vector direction as seen in Eq. (37) (ANSYS 13.0 Fluent, 2006). Note that more information and details of thermal comfort indices can be found in reference papers (Fanger, 1972), (International Organization for Standardization, 2007).

$$PMV = (0.03e^{-0.036M} + 0.028)\{(M - W) - 3.05 \times 10^{-3} \times [5733 - 6.99(M - W) - p_a] - 0.42 \times [(M - W) - 58.15] - 1.7 \times 10^{-5}M(5867 - p_a) - 0.0014M(34 - t_a) - 3.96 \times 10^{-8}f_{cl} \times [(t_{cl} + 273)^4 - (\bar{t}_r + 273)^4] - f_{cl}h_c(t_{cl} - t_a)\}$$
(19)

with

$$t_{cl} = 35.7 - 0.028(M - W) - c_{cl} \{3.96 \times 10^8 f_{cl} \times [(t_{cl} + 273)^4 - (\bar{t}_r + 273)^4] + f_{cl}h_c(t_{cl} - t_a)\}$$
(20)

$$h_{c} = \begin{cases} 2.38(t_{cl} - t_{a})^{0.25} \\ 12.2\sqrt{v_{ar}} \end{cases} \text{ for } \begin{cases} 2.38(t_{cl} - t_{a})^{0.25} > 12.1\sqrt{v_{ar}} \\ 2.38(t_{cl} - t_{a})^{0.25} < 12.1\sqrt{v_{ar}} \end{cases}$$
(21)

$$f_{cl} = \begin{cases} 1.00 + 1.290I_{cl} \\ 1.05 + 0.645I_{cl} \end{cases} \text{ for } \begin{cases} I_{cl} \le 0.078 \ (m^2.\,^{\circ}\text{C}/W) \\ I_{cl} > 0.078 \ (m^2.\,^{\circ}\text{C}/W) \end{cases}$$
(22)

where *M* is the metabolic rate (W/m^2) , *W* is the external work (W/m^2) (close to zero for most activities), p_a is the partial water vapour pressure (Pa), t_a is air temperature (°C), f_{cl} is the ratio of body's surface area (while clothed) over the surface area (while naked), t_{cl} is the surface temperature of clothing (°C), \bar{t}_r is the mean radiant temperature (°C), h_c is the convective heat transfer coefficient $(W/m^2.$ °C), v_{ar} is the relative air velocity (with reference to human body) (m/s) and I_{cl} is the thermal resistance of clothing $(m^2.°C/W)$.

The PPD index can be evaluated by the formula below as,

$$PPD = 100 - 95 \times e^{-(0.03353 \times PMV^4 + 0.2179 \times PMV^2)}$$
(23)

Table 2.1 gives the relation between the PMV indices and the thermal sensation conditions. Negative values show the feeling of coolness while positive values show the feeling of warmness.

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

Table 2.1. Relationship between PMV and thermal sensation

2.9. Theory and physical parameters and dimensionless variables

The following section describes the detailed process of solving and post-processing heat transfer and thermal environment problems.

2.9.1. Buoyancy-driven flow theory

Buoyancy-driven flow is a flow that is induced due to the force of gravity acting on the density variations when heat is added to a fluid and the fluid density varies with temperature. It is applied to natural convection and mixed convection flows. In mixed convection flow, the strength of buoyancy forces can be measured by the ratio of Grashof $(Gr = g\beta\Delta TL^3/v^2)$ and Reynolds (Re = vL/v) numbers:

$$\frac{Gr}{Re^2} = \frac{g\beta\Delta TL}{v^2}$$
(24)

where β is thermal expansion coefficient, ΔT is temperature difference, L is characteristic length (m) and v is velocity (m/s), v is kinematic viscosity (m²/s).

When $\frac{Gr}{Re^2} \gg 1$, then strong buoyancy may be contributed to the flow. When $\frac{Gr}{Re^2} \ll 1$, then buoyancy forces may be ignored. For the case of $\frac{Gr}{Re^2} \approx 1$, both natural and forced convection effects must be considered due to the presence of both buoyancy and inertia forces.

2.9.2. 2-D conjugate heat transfer model study

In 2-D case study on conjugate heat transfer, simulation results calculated in this study are compared with those obtained by other researchers (Kuznetsov and Sheremet, 2011) in the dimensionless physical variables (e.g. temperature and stream function) at dimensionless time (τ) as follows.

$$\theta = \frac{(T-T_0)k_{rad}}{q_0 L^2} \tag{25}$$

$$\Psi = \psi \sqrt{\frac{k_{rad}}{g \beta q_0 L^5}} \tag{26}$$

$$\tau = t \sqrt{\frac{g \beta q_0 L}{k_{rad}}}$$
(27)

where θ is dimensionless temperature, T_0 is initial temperature, k_{rad} is thermal conductivity of radiator panel, q_0 is volumetric thermal-power density of heat source (W/m^3) , L is streamwise length of fluid domain (m), Ψ is dimensionless stream function, ψ is stream function (m^2/s) , g is gravity (m^3/s) , β is thermal expansion coefficient, τ is dimensionless time and t is time (s).

The stream function (ψ in a unit of m^2/s) can be calculated using stream function in a unit of kg/s by ANSYS Fluent solver (Eq. (28)) divided by fluid density. The dimensionless stream function (Ψ) is then calculated using Eq. (26) described above.

$$\rho u \equiv \frac{\partial \psi}{\partial y}, \rho v \equiv \frac{\partial \psi}{\partial x}$$
(28)

where u and v are velocity component.

Other physical parameters used throughout the 2-D CHT study are

$$Pr = \frac{v}{a} \tag{29}$$

$$Gr = \frac{g \beta q_0 L^5}{v^2 k_{rad}}$$
(30)

where Pr is Prandtl number, ν is momentum diffusivity, α is thermal diffusivity (m^2/s) , Gr is Grashof number and ΔT is temperature difference.

2.9.3. 3-D conjugate heat transfer model study

For a large plane wall, one-dimensional (1-D) heat conduction equation can be applied and using the Fourier's law, the equation can be written as

$$Q_{cond} = -k A \frac{dT}{dx}$$
(31)

where Q_{cond} is conductive heat transfer, k is thermal conductivity of solid material, dT/dx is the temperature gradient, and A is the heat conduction area (m^2) . Thus, the total and surface heat fluxes can be evaluated by

$$q_{total} = \frac{T_{in,\infty} - T_{out,\infty}}{R_{total}}$$
(32)

$$q_{surface} = h_{in \ or \ out} \left(T_{in \ or \ out, \infty} - T_{inner \ or \ outer \ surface} \right)$$
(33)

where q_{total} is total heat flux (W/m^2) , R_{total} is total thermal resistance (i.e., *R*-value) (m^2K/W) , $q_{surface}$ is surface heat flux, *h* is heat transfer coefficient (W/m^2K) and indexes *in* and *out* are internal and external environments, subscript ∞ is ambient condition and *inner surface* and *outer surface* are internal and external surfaces of a finite-thickness wall.

An energy balance over a wall thickness of Δx within a small time interval (i.e. before thermal equilibrium fully established) can be expressed as

$$\frac{1}{A}\nabla\left(k\,A\,\nabla T\right) + \,\dot{e}_{gen} = \frac{1}{\Delta t}\left(\rho\,c_{p}\nabla T\right) \tag{34}$$

where \dot{e}_{aen} is heat generation per unit volume (W/m^3) .

By considering a constant thermal conductivity (which is generally valid for most practical applications), steady-state heat transfer and no extra heat generation inside the solid domain, Eq. (34) can be further simplified to a Laplace equation of temperature (Eq. (6)).

By defining proper boundary conditions at computational domain boundaries, this Laplace equation can be discretised and solved in a straightforward manner, resulting the conduction heat as a linear function of streamwise position, i.e. T = mx + n, where constant parameters (m and n) are determined by boundary conditions.

2.9.4. 3-D thermal comfort study

The comfort temperature is another variable to describe the occupant's feeling of the thermal climate in a room environment, which considers the balance of radiation and convection heat transfer modes, and it can be used for results comparison with available data from other published sources (e.g. (Myhren and Holmberg, 2008), (Myhren and Holmberg, 2009)).

$$T_{comfort} = \frac{T_{radiation} + T_{air}\sqrt{10 U}}{1 + \sqrt{10 U}}$$
(36)

where $T_{comfort}$ is comfort temperature, $T_{radiation}$ is radiation temperature (ANSYS 13.0 Fluent, 2006) below, T_{air} is air temperature and U is air velocity magnitude.

$$T_{radiation}^{4} = \frac{1}{4\sigma} \int_{0}^{4\pi} I \, d\Omega \tag{37}$$

where Ω is solid angle.

To determine air velocity at a ventilation opening slot location based on either given ventilation rate (m^3/s) or air supply rate (L/s), following equations can be used.

Air supply rate
$$(L/s) = \frac{Air \ change \ rate \ (1/h) \times Room \ volume \ (m^3) \times 1000 \ (L/m^3)}{3600 \ (s/h)}$$
 (38)

Air velocity
$$(m/s) = \frac{Ventilation rate (m^3/s)}{Cross-sectional area of inlet opening (m^2)}$$
 (39)

2.10. Summary

Airflow and heat transfer performance of built environment has been an interesting research subject. The physics of heat transfer in enclosed space is very complex since the coupling of fluid flow and heat transfer would be complex and challenging even for a single natural convection model room. This is due to the nonlinearity of the physical problem itself and also the interactions between the closely related flow field and temperature field. The investigation in this thesis employed a commercial CFD code to simulate and analyse characteristics of airflow and heat transfer in a closed space with furniture and occupant and to optimise indoor thermal environment. In this chapter, the mathematical models with the governing equations, which describe the heat transfer in convection, conduction and radiation modes, are presented. The employed physical parameters, solver and computational grids used for airflow and heat transfer are also provided. The mathematical models of flow unsteadiness (i.e. FFT techniques) and thermal comfort (i.e. Fanger's indices) are summarised and adopted in Matlab and FORTRAN programmes, respectively. In addition, the key factors for this thesis, such as theory of buoyancy-driven flow, simplified conjugate heat transfer model and radiation temperature equation, are also summarised.

Chapter 3

Benchmark Cases

The aim is to present a comprehensive benchmark study of indoor airflow and thermal field for convectional and buoyant flow features in two-dimensional (2-D) empty model room (three different configurations), and three-dimensional (3-D) model room with and without a built-in box (obstacle) of non-heated or heated source (three different scenarios in a basic configuration). CFD results will be validated against available experimental data in terms of air velocity, kinetic turbulence energy and temperature distributions. The focuses in the 2-D model study will be the mesh topology and turbulence model effects. The more complicated flow features will be studied for a 3-D model room with a box embedded in the room with interests of the effect of different solver and model, and boundary condition.

3.1. Two-dimensional flow and heat transfer benchmark cases

Indoor heat transfer analysis was numerically studied using three different twodimensional models in three different convective heat transfer modes, see Table 3.1. The cases are simple configurations such as a square- or rectangle-shape without obstacle inside fluid domain. Simple boundary conditions are applied to each case, e.g. ventilated incoming airflow or heated/cold walls.

Case	Heat transfer mode	Model	Consideration
3.1.1	Forced convection	A horizontally rectangular ventilated domain	Mesh topology
3.1.2	Natural convection	A vertically rectangular closed domain with a heated wall	Turbulence model
3.1.3	Mixed convection	A square ventilated domain with a heated floor	

Table 5.1. Summary of 2-D benchman	ark (cases
------------------------------------	-------	-------

For case 3.1.1, additional unstructured grid and hybrid grid were also generated for results comparison. For case 3.1.2, two-equation turbulence models, the re-normalized group (RNG) $k - \varepsilon$ model and the shear stress transport (SST) model are adopted to take account of the Reynolds stress and turbulent heat flux for momentum and energy equations. For each case studied here, a careful grid convergence study was performed using Gambit software. The CFD predicted results will be validated against available experimental data in terms of air velocity and temperature distributions in normalized form by focusing on the mesh topology and turbulence model effects in the 2-D study.

3.1.1. Forced convection in a 2-D empty room (Case 3.1.1)

The forced convection is a way of transport heat/energy using an external device such as apump, fan, etc. In this scenario, the buoyancy effect is relatively small compared to the kinematic movement of the fluid, thus often be negligible. Figure 3.1 gives a sketch of 2-D model room of a height H = 2.87 m and a width W= 3H, following Restivo's experiment (Restivo, 1979), with ventilation airflow coming in through an inlet slot of a narrow height 0.056H near the ceiling on the left and going out from an outlet slot of height 0.16H at the floor on the right. The Reynolds number is estimated 5,000, based on the inlet slot height and the inlet air velocity of 0.455 m/s and temperature of 15 °C. Hence the flow in the room can be assumed to be turbulent, based on this Reynolds number (also see studies by Zuo and Chen (Zuo and Chen, 2009)). Present simulation considers the RNG k- ε turbulence model with an inlet turbulence intensity of 5.5 %. CFD predicted streamwise component velocity (U_x) profiles, at two locations of x = H and 2H, are compared to the experimental data of Restivo (Restivo, 1979).



Figure 3.1. Sketch of forced convection airflow in a 2-D model room.

The boundary conditions applied are summarized in Table 3.2; uniform air velocity of 0.455 m/s at inlet, no mechanical outflow condition representing the assumptions of zero pressure gradients at outlet and overall mass balance inside domain, and non-slip adiabatic wall conditions for wall surfaces.

0.455 m/s
No mechanical outflow
Adiabatic condition

Table 3.2. Boundary conditions of 2D forced convection study

For mesh topology study, the computational domain was decomposed to three regions; an upper region of the inlet flow slot, a lower region of the outlet flow slot and a middle region of the remaining room in order to apply three different mesh topologies at similar grid density and resolution, e.g. a fully structured mesh, a fully unstructured mesh and a mixed mesh which keeps structured grids in the upper and the lower regions, while applying unstructured grid in the middle region.

A grid of 36×36 , non-uniformly distributed across the domain with finer meshes clustered in inlet and outlet, and near wall regions was produced using Gambit software. The number of grid points and their distributions in each region was close to that adopted by Zuo and Chen (Zuo and Chen, 2009), and the grid size differential at the interface between two regions was minimized by carefully tuning the grid stretch factor. It was found that both structured and mixed mesh results show good agreement with the measurements, while the unstructured grid gives poor predictions, particularly in the near wall region as shown in Figure 3.2.

For grid refinement study with a structured mesh, the number of grid points in x and ydirections was increased by a factor of 1.5 (i.e. 54×54 grids) and a factor of 2 (i.e. 72×72 grids), respectively. The results showed a consistent prediction among three grid resolutions and were in good agreement with the experiment data. Thus the graphs of the results are excluded here but are found in Horikiri et al. 2011 (Horikiri et al, 2011). Comparing results from 36×36 and 54×54 grids, the differences of root-mean-square (RMS) velocity are merely $0.04 \ m/s$ at x = 1H, and $0.05 \ m/s$ at x = 2H, respectively, and by refining to a 72×72 grid they further reduce to 0.01 m/s for x = 1H and 2H. It was found that the x-refinement has negligible influence on predicted velocity distributions, while the y-refinement did slightly improve the predictions.



Figure 3.2. Comparison of x-component velocity at (a) x = H and (b) x = 2H on mesh topology, compared with experiment (Restivo, 1979).

3.1.2. Natural convection in a 2-D tall cavity (Case 3.1.2)

Natural convection is another way of heat transfer that uses buoyancy force to drive airflow movement. It does not need any external force and a representative case is the winter air circulation in a room enclosure when all ventilation slots are closed to prevent the heat loss. The natural convection of airflow in a tall cavity was considered, which presents basic flow features in buildings. The experimental study was done by Betts and Bokhari (Betts and Bokhari, 2000) and the case was also numerically studied by Zuo and Chen (Zuo and Chen, 2009) using FLUENT software.

The domain has a width of 0.076 *m* and a height of 2.18 *m*, as shown in Figure 3.3. The left wall was kept 'cold' at a fixed temperature of $T_{cold} = 15.1$ °C and the right wall was kept 'warm' at a fixed temperature of $T_{hot} = 34.7$ °C, while the top and the bottom walls were both insulated (i.e. zero heat flux at wall surfaces), respectively. The Rayleigh number is 0.86×10^6 , based on the domain width, and this corresponds to a turbulent flow as discussed by Betts and Bokhari (Betts and Bokhari, 2000). Hence the study considered two turbulence models, RNG $k - \varepsilon$ model with enhanced wall treatment option and SST $k - \omega$. The summary of boundary conditions is in Table 3.3.

Initial study used a non-uniform structured mesh of 20×10 , following the work of Zuo and Chen. Both grid refinements by a factor of 1.5 and 2 (i.e. 30×15 , 40×20 grids); and also by gradient-based dynamic mesh adaption were performed, the SST $k - \omega$ model resulting in a non-uniform structured mesh of 5,800 grid points for final simulation. In the result analysis, vertical velocity component and temperature were compared in dimensionless form, i.e. $U_y = u_y/u_{max}$ and $\theta = T/T_{max}$ at three different heights, Y = 0.1, 0.5 and 0.9 with respect to turbulence model effect. A maximum velocity for the calculation was 0.19 m/s obtained through the case. The minimum and maximum temperatures are the cold and hot wall temperatures respectively.



Figure 3.3. Sketch of the natural convection flow in a 2-D room

Table 3.3. Boundary	conditions of 2D	natural	convection	study
---------------------	------------------	---------	------------	-------

Right wall	34.7 °C
Left wall	15.1 °C
Other walls	Adiabatic condition

Figure 3.4 shows vertical velocity and temperature profiles at three horizontal locations of Y = 0.1, 0.5 and 0.9 along the cavity height from grid adaption simulation,

in comparison with test data of Betts and Bokhari (Betts and Bokhari, 2000). It can be seen that while both turbulence models predicted similar trends, the SST model predicts better results in agreement with test data than those by the RNG $k - \varepsilon$ model, particularly near wall boundary layer profiles. This is partly because that the *SST* $k - \omega$ model is capable to model large turbulent levels in regions with strong velocity developments. There are some discrepancies in velocity and temperature profiles between the CFD prediction and the experimental measurement, seen at two locations Y = 0.1 and 0.9 close to the upper and lower walls. The similar observations were shown in a previous study by Zhang et al. (Zhang et al, 2007), using various turbulence models. Although the initial grid of 20 × 10 has very high mean aspect ratio of 14, ANSYS FLUENT software is still capable to produce reasonably good results (Horikiri et al, 2011) and after the mesh adaption with increased mesh density and decreased mean aspect ratio, results presented here have shown some improvements in velocity near the walls and temperature inner side of the domain between X = 0.1 and X = 0.9 at locations close to upper/lower walls.



Figure 3.4. Comparison of vertical velocity (left) and temperature (right) profiles at Y = 0.1, 0.5 and 0.9 in different turbulence models compared with experiment (Betts and Bokhari, 2000).

3.1.3. Mixed convection in a 2-D empty room (Case 3.1.3)

The mixed convection mode represents a combination of forced and natural convections. This study considered a mixed convective airflow experiment done by Blay et al. (Blay et al, 1992) using a sqaure cavity with a heated bottom wall with inlet and outlet ventilation slots and numerically studied by Chen (Chen, 1996).

Figure 3.5 shows a 2-D configuration of 1.04 *m* edge-length (H = W), in which an airflow of 0.455 *m/s* velocity is introduced at the front wall through a 0.018 *m* height inlet-slot near the ceiling and an open exit of 0.024 *m* height from the floor located at the back wall. The front and back walls and the upper wall are kept at a lower constant temperature (15 °C), same as the inlet airflow temperature, while a higher temperature (35 °C) is kept for the bottom wall, inducing a buoyancy effect, summarized in Table 3.4. The strength of buoyancy driven effect was measured by a ratio of Grashof (*Gr*) number to square of Reynolds (*Re*) number (*Gr/Re*²) and this was estimated 0.06 at 'mean' temperature of 25 °C, which is an averaged value of upper and bottom walls. Hence in this case the buoyancy effect is negligible compared to the air convection.

The computational grids were 20×20 structured mesh in non-uniformly distribution, similar to that of Chen's study (Chen, 1996). Although noting that the RNG $k - \varepsilon$ model generally produces better predictions for a fully-developed high Reynolds number turbulent flow, a RNG $k - \varepsilon$ turbulence model was used for this case after considering the results of previous two cases. The CFD predicted results were validated with available experiment data in the dimensionless form of velocity (U) and temperature (θ) at X = 0.5 throughout the room heigh. The maximum velocity used for the calculation was 0.455 m/s, i.e. the velocity at inlet. The temperatures of hot surface and cold surface of the domain were the maximum and minimum temperatures in the dimensionless form.



Figure 3.5. Sketch of the mixed convection flow in a 2-D room.

Table 3.4. Boundary conditions of 2D mixed convection study

Inlet	0.455 m/s
Outlet	No mechanical outflow
Bottom wall	35 ℃
Other walls	15 ℃

Figure 3.6 depicts velocity vectors and temperature contour showing airflow direction, flow pattern variations and temperature distribution. The velocity vectors show the air movement with a large circulation around the geometrical centre of the room, and higher velocities are seen in the outer circulation flowing along the walls. A thermal plume was observed around the corner of the front and bottom walls while more cold air moved to the inner domain with a help of forced inflow.

The predicted velocity and temperature profiles are compared with the Blay's experimental data with reasonably good agreements shown in Figure 3.7. Despite that a merely 20×20 grids were used in the lower to intermmediate Reynolds number turbulent flow, the RNG $k - \varepsilon$ model were able to produce fairly good results, as previously studied by other researchers (Zuo and Chen, 2009), (Zhang et al, 2007). Further refining the grid by a factor of 1.5 (i.e. 30×30) and 2 (i.e 40×40), RMS differences are merely 0.02 m/s for velocity and $0.5 \,^{\circ}$ C for temperature, respectively. Compared to the measurements, CFD predicted temperature adjacent to the floor and the inlet was underestimated by $3 \,^{\circ}$ C and $0.7 \,^{\circ}$ C, respectively, and the inlet velocity was overestimated

by approximately 0.04 m/s, this resulted in computed Froude number (*Fr*) of 4.47, slightly higher than measured Froude number of 4.18 by Blay et al. (Blay et al, 1992).



Figure 3.6. Velocity vector (a) and temperature (b) distributions of 2-D mixed convection flow.



Figure 3.7. Comparison of velocity (a) and temperature (b) profiles at X = 0.5 with experiment (Blay et al. 1992).

3.1.4. Summary of 2-D benchmark case studies

The 2-D benchmark case studies show that the numerical models developed here were simulated using ANSYS Fluent. The CFD predicted results were validated against available experimental data, showing good agreement in air velocity and temperature distributions. It was found that structured mesh gave better resolution in the near wall region than unstructured mesh, leading good agreement with the measurements. The turbulence model study shows that both turbulence models generally predicted flow features in good agreement with experiment. Particularly the *SST* turbulence model predicted better the flow details near wall boundary layer profiles in low turbulent Reynolds number flow.

3.2. Three-dimensional flow and heat transfer benchmark cases

The model considered here was based on 3-D model room experiment studied by Wang and Chen (Wang and Chen, 2009). The cases were gradually complex in terms of fluid mechanism and configuration from empty room to room with and without a nonheated and heated box located at the centre of a square domain (see Table 3.5). Airflow in 3-D model rooms represents most heat and mass transfer scenarios in an enclosed indoor environment. The focus of the present study is on three-dimensional flow characteristics compared with two-dimensional flow studied earlier, flow pattern caused by the presence of a non-heated box as well as velocity and temperature distributions, and formation of thermal plume and buoyancy flow from a heated-box in the centre of the domain.

Case	Model	Consideration
3.2.1	Forced convection in an empty square room	-
3.2.2	Forced convection in a square room with unheated cubic box	Solver and Turbulence model
3.2.3	Mixed convection in a square room with heated cubic box	Boundary condition

Table 3.5. Summary of benchmark cases in three-dimensional

A cubic test room has an edge length of 2.44 m (H = W = L) with an inlet slot (a height of 0.03 m attached to the ceiling across the whole width) at the front wall and an exit slot (a height of 0.08 m attached to the floor across the whole width) at the back wall, as seen Figure 3.8. In Case 3.2.2, a cubic box with an edge length of 1.22 m is located at the centre of the room (see Figure 3.8) and the box generates a uniform heat of 700 Watts in Case 3.2.3, which is equivalent to 36.7 °C uniformly distributed on the box surfaces. Through the cases, the incoming air is applied with a flow rate 0.10 m^3/s and a temperature of 22.2 °C. Based on physical condition of the heated box, i.e. $Gr/Re^2 \ll 1$, and $Re_{inlet} > 2000$, where Gr is Grashof number, Re is Reynolds number, the heat transfer due to forced convection mode will play a major role in the heat transfer process, compared to natural convection mode. The domain wall surfaces are adiabatic.

The problems are solved by finite volume numerical method on a structured grid. In the near-wall regions, fine mesh distribution was applied to capture both flow and thermal boundary layars, $30 < y^+ < 50$. A grid independent study is conducted using two successive grid resolutions of $36 \times 36 \times 36$ and $54 \times 54 \times 54$ grid points nonuniformly distributed in the Cartesian coordinates (x, y and z) and a grid independent solution was achieved with $36 \times 36 \times 36$.

The 'mean' results were calculated based on 20 sub-datasets, each averaging over 5,000 iterations/time-steps during the calculation. Note that unsteady-state mode calculates with a time step of 0.055 s determined by a smallest mesh size divided by the inflow velocity. Averaged results from unsteady-state calculation were compared in velocity, turbulence kinetic energy and temperature in normalised form (i.e. U, K and T) with available experimental data of Wang and Chen (Wang and Chen, 2009). The velocities were normalized by the maximum velocity, $u_{max} = 1.5 m/s$ and the turbulence kinetic energy normalized by the maximum of $k_{max} = 0.05 m^2/s^2$. For temperature in case 3.2.3, an equation of $(T - T_{min})/(T_{max} - T_{min})$ was used with $T_{min} = 22.2 \ ^{\circ}C$ i.e. air temperature at inlet slot, and $T_{max} = 36.7 \ ^{\circ}C$, i.e. temperature of heated box, as reference values. The results were shown only at five monitoring/ measuring positions for each case, illustrating the level of accuracy compared with the measured data and the characteristics of flow distributions in terms of velocity, turbulence energy and temperature.

The results are compared in air velocity, turbulence kinetic energy (TKE) and temperature with available experiment data on a streamwise and a cross-sectional midplanes, and at 10 monitoring positions (shown in dot mark in Figure 3.8(b) and Table 3.6 for the coordinates), by focusing on the effects of different turbulence models, solvers and boundary conditions. The solid line in Figure 3.8(b) represents the position of an internal box for cases 3.2.2 and 3.2.3.



Figure 3.8. Sketch of a 3-D model room with dimensions and inlet/outlet slot (a) and ten measuring positions along two central lines in horizontal x-z plane (b).

Point	Coordinates (X, Z)	Point	Coordinates (X, Z)
1	(0.09, 0.5)	6	(0.5, 0.09)
2	(0.28, 0.5)	7	(0.5, 0.28)
3	(0.47, 0.5)	8	(0.5, 0.47)
4	(0.66, 0.5)	9	(0.5, 0.66)
5	(0.84, 0.5)	10	(0.5, 0.84)

Table 3.6. Location of monitoring points

3.2.1. Forced convection in a 3-D empty room (Case 3.2.1)

Figure 3.9 and Figure 3.10 show CFD predicted velocity vectors/contours and turbulence kinetic energy contours at 20,000 iterations in normalised form, in comparison with the experimental data (Wang and Chen, 2009) on a streamwise mid-plane (x, y, z =L/2) and a spanwise mid-plane for case 3.2.3. The CFD results in contour plots were obtained by ANSYS FLUENT using the RNG k- ε model. The predicted flow structures were qualitatively in good comparison with the measurement with more complicated patterns developed by additional box (obstacle) inside the domain. The existence of an unheated box (i.e. case 3.2.2) induced two air recirculation regions in clockwise, one above the top surface of the box and the other between the box rear surface and domain right wall. The recirculations are shown in dark-blue of velocity contour, indicating lower velocities in the centre of flow circulation. In case of heated box (i.e. case 3.2.3), CFD prediction shows a large anti-clockwise recirculation forming above the heat source, which was not observed in the experiment. This was probably due to less stability in flow current caused by weak buoyant formation above the heated box as seen in Figure 3.13, resulting in large temperature difference between the fluid adjacent to the heated box and the fluid away from it. At position 5, the difference between the predicted instantaneous velocity and the measurement was as large as 29.4 % for case 3.2.2 and 51.9 % for case 3.2.3. The difference between a time-averaged velocity magnitude and the test data however was decreased dramatically to 6 % for case 3.2.2 and 5 % for case 3.2.3, respectively. The CFD predicted turbulence kinetic energy was also overestimated in the ceiling region. For case 3.2.2 and case 3.2.3, due to an obstacle placed in the flow domain, the turbulence kinetic energy between the domain right wall and the box is high. The discrepancies between instantaneous prediction (at 20,000 iterations) and the experiment data were very large about 34 % and 26 % difference in the ceiling region in case 3.2.2 and case 3.2.3 respectively. The discrepancies were reduced dramatically when comparing timeaveraged prediction with the test data shown below.



Figure 3.9. Comparison of normalized velocity magnitude (U) vectors/contours between experiment (top) and CFD prediction (bottom) at a middle plane (x, y, z = 0.5L).



Figure 3.10. Comparison of normalized turbulence kinetic energy (K) contours between experiment (top) and CFD prediction (bottom) at a middle plane of (x, y, z = 0.5L)

Figure 3.11 gives time-averaged velocity and TKE results in comparison with test data for case 3.2.1. It can be seen that the RNG turbulence model captured velocity and turbulence kinetic energy more accurate than the SST model, compared with experiment. In which case, the averaged velocity results of the RNG turbulence model were generally in good agreement with measurement. The largest discrepancies occurred at position 5, where the airflow has strong curvature in flow direction. The time-averaged TKE predictions also showed similar trend compared with the experiment, although TKE is often associated with greater uncertainties and error in both CFD and experiment as previously discussed (Chen and Srebric, 2001). The overestimations near the ceiling region may be due to high local turbulence intensity as discussed earlier.



Figure 3.11. Comparison of normalised velocity and turbulence kinetic energy distributions at positions 2, 5 and 8 (case 3.2.1) with experiment (Wang and Chen, 2009).

3.2.2. Forced convection in a 3-D room with a unheated box (Case 3.2.2)

Figure 3.12 shows the comparison of CFD predictions and the experimental measurements (Wang and Chen, 2009) for case 3.2.2. Two solvers in ANSYS suite (CFX and FLUENT) with $k - \varepsilon$ turbulence model were applied. The comparison of normalised velocity and TKE profiles showed broadly good agreement, except for position 5 where in CFX with RNG $k - \varepsilon$ model showed under-prediction through the domain height. In contrary, both FLUENT RNG $k - \varepsilon$ model results and CFX standard $k - \varepsilon$ model results were in good agreement with measured data. For consistency, FLUENT solver was continued in use for case 3.2.3.



Figure 3.12. Comparison of normalised velocity and turbulence kinetic energy distributions at positions 2, 5 and 8 (case 3.2.2) with experiment (Wang and Chen, 2009).

3.2.3. Mixed convection in a 3-D room with heated box (Case 3.2.3)

Figure 3.13 shows the instantaneous temperature contours at 20,000 iterations at streamwise and spanwise mid-planes using a fixed wall temperature for heated-box using FLUENT RNG model, in comparison with experiment in normalised form. It shows that the inflow temperature remained cooler due to weak thermal plume generated by the heated-box in CFD results. Thermal stratification was seen on the rear wall of the heated-box (Figure 3.13(c)).



Figure 3.13. Comparison of predicted temperature contours at 20,000 iterations (bottom) on a streamwise (a,c) and spanwise (b,d) mid-planes with experiment (Wang and Chen, 2009) (top) (case 3.2.3)

The effect of thermal condition of heated-box in the domain was studied using FLUENT RNG $k - \varepsilon$ model and results were shown in Figure 3.14. In case of applying fixed wall heat flux for the heated-box, the experimental 700 W heat uniformly generated over the centred box was equivalent to 94.06 W/m^2 of surface heat flux. Otherwise the box

surfaces had a uniform temperature surface of 36.7 °C. Simulations were run for both thermal conditions for heated-box case 3.2.3. The time-averaged velocity and turbulence kinetic energy have shown insignificant discrepancies between these two conditions, whereas the temperature did show over-predictions for the heated-box with the fixed heat flux condition. For temperature distribution comparisons, results using fixed wall temperature for heated-box were in better agreement with the experimental data at all 10 monitoring/measuring positions.



Figure 3.14. Comparison of normalised velocity, turbulence kinetic energy and temperature predictions at positions 1, 6 and 8 (case 3.2.3) with experiment (Wang and Chen, 2009).

3.2.4. Summary of 3-D benchmark case studies

Based on above analysis of 3-D heat transfer studies with the effect of turbulence model, solver and boundary condition, it can be seen that the three different cases in different heat transfer mode were validated against experiment and the results were influenced by turbulence model, solver and boundary conditions. The good agreement in velocity, TKE and temperature of the 3-D heat transfer cases were obtained in ANSYS Fluent RNG $k - \varepsilon$ turbulence model with temperature boundary condition on the heated internal box, compared with experiment.

3.3. Flow unsteadiness in a 3-D benchmark case

The investigation of flow unsteadiness analysis is conducted using the 3-D benchmark cases (in section 3.2), concentrating on flow oscillation phenomena, for which the time history of velocity and temperature date will be analyzed using Fast Fourier Transformation (FFT) technique. This section will develop the detailed information of flow unsteadies in a domain and assess the cause of flow instability raised in the previous study (Horikiri et al, 2012).

The time-history of velocity magnitude and temperature (only case 3.2.3) at monitoring/measuring points were stored every 10 iterations in order to analyse the flow unsteadiness, using the Fast Fourier Transform (FFT) technique. The results are computed by Matlab based on the time-history data only at five monitoring locations P1, P3, P5, P6 and P10 at five different heights, Y = 0.1, 0.3, 0.5, 0.7 and 0.9 from the floor.

3.3.1. Oscillation signal

Figure 3.15 shows velocity and temperature fluctuation in time period (100 - 500 s) at Y = 0.5 of P10 for three cases. Based on the time step and the velocity at the inlet slot, i.e. 1.378 m/s, the time period is equivalent to 10 - 70 complete flow circulations. It can be seen that there is no particular trend or similarity in fluctuation variation among three cases and the amplitude of the spikes is asymmetry. The fluctuation variation at the location in case 3.2.1 seems quieter, e.g. 0.025 - 0.075 m/s in fluctuation range, than

that of case 3.2.2 and case 3.2.3, for which the range of velocity is 0.01 - 0.15 m/s. The presence of unheated box in the domain (case 3.2.2) causes high frequency of velocity oscillations and wide range of velocity magnitude. The high frequency of noise is slightly moderated in case 3.2.3, with a heated box and a new frequency of velocity oscillation seems to be consistent to that of temperature oscillation.



Figure 3.15. Time serials of velocity magnitude and temperature at Y = 0.5 at P10 for three cases.

3.3.2. Power spectral density (PSD) analysis

Figure 3.16 and Figure 3.17 show power spectral density (PSD) of velocity magnitude in time series at five monitoring locations for an empty domain (case 3.2.1). It can be seen that the base frequencies are found lower than 0.2 Hz (Tian and Karayiannis, 2000) and the maximum power spectral density is 2.0 m^2/s at Y = 0.3 of P6. A single dominant frequency is found 0.17 Hz at Y = 0.1 at P1 and Y = 0.1, 0.3 and 0.9 at P5, and 0.05 Hz - 0.17 Hz through the height of P6 location. In general, the large energy of velocity fluctuations is found near floor and ceiling levels at P5 (along the domain back

wall), P6 and P10 (on a spanwise plane) while the energy spectra is particularly small near the inlet opening (e.g. Y = 0.9 at P1 and P3) and at the mid-height of the domain (Y = 0.5) (Tian and Karayiannis, 2000).



Figure 3.16. Energy spectra of time-history velocity magnitude fluctuations at monitoring locations *P*1, *P*3, *P*5 for case 3.2.1.



Figure 3.17. Energy spectra of time-history velocity magnitude fluctuations at monitoring locations *P*6, *P*10 for case 3.2.1.

Figure 3.18 and Figure 3.19 show power spectral density of velocity magnitude in time series at five monitoring points for an unheated box (case 3.2.2). It can be seen that the base frequencies are lying between 1.0 Hz and 5.0 Hz. The maximum power spectral density of 1.25 m^2/s and a double frequency (e.g. at 2.15 Hz and 4.3 Hz) are observed at Y = 0.1 of P6. The range of active frequencies is higher and maximum energy of the velocity oscillations is lower than those in case 1. The energy of velocity fluctuation increases with the distance from the ceiling level (i.e. Y = 0.9 level) for the monitoring location P1, P6 and P10. However its energy is particularly weak in the upper levels of the domain for all the monitoring locations. The dominant frequency is clearly seen at different magnitudes in the lower levels of 0.1 < Y < 0.5 on a streamwise mid-plane whereas at 2.15 Hz for P10 in 0.1 < Y < 0.3 on a spanwise mid-plane). At P3, due to a clockwise recirculation on the top of the box and inflow jet near the ceiling, the behaviour of flow oscillations is irregular between level Y = 0.7 and level 0.9, where a dominant frequency is in the range of 0 - 1.0 Hz and 4.0 - 5.0 Hz, respectively.



Figure 3.18. Energy spectra of time-history velocity magnitude fluctuations at monitoring locations *P*1, *P*3, *P*5 for case 3.2.2.



Figure 3.19. Energy spectra of time-history velocity magnitude fluctuations at monitoring locations *P*6, *P*10 for case 3.2.2.
Figure 3.20 - Figure 3.23 show power spectral density of velocity magnitude and temperature fluctuations in time series for a heated box in the fluid domain (case 3.2.3). It can be seen that the base frequencies for velocity and temperature fluctuations lie below 5.0 Hz although there is large energy difference between velocity and temperature fluctuations. The maximum power spectral density is 6.0 m^2/s at Y = 0.1 of P10 for velocity, stronger than that of case 3.2.2 and 85.0 m^2/s at Y = 0.5 of P10 for temperature. The dominant frequencies in the velocity oscillation are consistent with those of temperature, around 1.3 Hz for P1 and P5 on a streamwise plane and 0.75 Hz for P6 and P10 on a spanwise plane, in which the trend that a different and higher dominant frequency is obtained at monitoring locations P1, P3, P5 on a streamwise midplane, compared with that of monitoring locations P6, P10 on a spanwise mid-plane is similar to case 3.2.2. Unlike case 3.2.2, there is no clear energy increase with domain height for velocity and temperature oscillations except for the P1 location. The highest energy of velocity oscillation at a location is found at Y = 0.1 level for the locations P1. P6 and P10, of which at P6 and P10, the power spectral density is three-time stronger than that of P1 (although the graphs show the Y-axis up to $1 m^2/s$. The strongest energy of temperature fluctuation is observed at P6 and P10 on a spanwise plane through the domain height, 2-3 times stronger than the highest energy value of the other locations (in Figure 3.22 and Figure 3.23). Furthermore the dominant frequencies of case 3.2.3 are lower than that of case 3.2.2, representing that the heat from the box increases flow steadiness in the fluid domain. This is probably due to the formation of thermal plume in the fluid domain, causing to stabilise the upper part and the sides of the heated box on a spanwise plane (see Figure 3.13) and also adiabatic wall condition in present study, conductive walls decreases flow stability (Henkes and Hoogendoorn, 1990).



Figure 3.20. Energy spectra of time-history velocity magnitude fluctuations at monitoring locations *P*1, *P*3, *P*5 for case 3.2.3.



Figure 3.21. Energy spectra of time-history velocity magnitude fluctuations at monitoring locations *P6*, *P*10 for case 3.2.3.



Figure 3.22. Energy spectra of time-history temperature fluctuations at monitoring locations *P*1, *P*3, *P*5 for case 3.2.3.



Figure 3.23. Energy spectra of time-history temperature fluctuations at monitoring locations *P6*, *P10* for case 3.2.3.

3.3.3. Summary of flow unsteadiness

The investigation of flow oscillation in 3-D ventilated model room has been carried out by computational fluid dynamics approach. The validated mathematical models against published data (Wang and Chen, 2009) have been used to investigate the development of flow unsteadiness in complex flow and thermal features created by adding a cubic box with/without thermal load in a domain. The results in forced convection flow in an empty-square domain (case 3.2.1) showed that there is no direct relation between velocity and turbulent flow in power spectral density and frequency, and each of time-history velocity oscillations is independent and random. The base frequency of velocity fluctuation is below 0.2 Hz. The energy of the velocity fluctuation is relatively weak at the mid-height of the domain (Y = 0.5). Adding a unheated box in the centre of the domain (case 3.2.2) induced more flow unsteadiness in the lower levels 0.1 < Y < 0.5 on a streamwise mid-plane (i.e. at P1 and P5) and at Y = 0.1 on a spawnwise mid-plane (i.e. at P6 and P10). A dominant frequency was observed larger than that of case 3.2.1 and its energy level was weaker, confirming that velocity oscillates faster. The dominant frequency depended on the orientation of the flow circulation, for example the monitoring locations P1 and P5 on a streamwise mid-plane had higher dominant frequency than the locations P6 and P10 on a spanwise mid-plane. For location P6, a double frequency relationship was found. The effect of a non-heated box in the domain on flow feature was seen on irregular oscillations in the level of 0.7 < Y < 0.9 of P3 where the strongest oscillation energy was found, caused by the clockwise recirculations above the top of the box. In case 3.2.3, the frequency of velocity oscillation was decreased compared with that of case 3.2.2 and its values were consistent with temperature at the location although the energy of the fluctuation is much higher in temperature. Similar to case 3.2.2, the oscillation dependency on flow orientation was seen in case 3.2.3. The strong energy of the oscillations is found at the lowest level (i.e. Y = 0.1) at P1, P6 and P10 for velocity and through the domain height at P6 and P10 (i.e. spanwise mid-plane) for temperature. It can be concluded that the formation of thermal plume from the heated box stabilised flow in the upper part and the sides of the heated box on a spanwise plane.

3.4. Summary of airflow and heat transfer study

The benchmark cases of airflow and heat transfer have been studied in 2-D and 3-D model rooms using ANSYS Fluent. Convective heat transfer cases were investigated with different numerical approaches (e.g. mesh topology, turbulence model, solver, etc) and validated against available experimental data. The results of each benchmark case showed good agreement with available data. It was found from the 2-D benchmark cases that air velocity increased along the boundary walls and especially hot wall which led flow direction upwards. At the centre of the flow circulation, air momentum is very weak (e.g. almost zero velocity magnitude). In terms of effect of numerical method, structured mesh gave better resolution in the near wall region than unstructured mesh, leading good agreement with the measurements. The turbulence model study shows that both turbulence models generally predicted flow features in good agreement with experiment. Particularly the SST turbulence model was capable to capture well the flow details near wall boundary layer profiles in low turbulent Reynolds number flow. Also the grid resolution in the ventilated areas controls the accuracy of the results. From the 3-D benchmark cases, it can be concluded that the increase of complex features (e.g. a box with/without heat) in the domain would lead to flow separations causing recirculations above the box and in the rear space of the domain and swirls in the front space presenting three-dimensional flow, and a thermal plume, compared with a two-dimensional clockwise flow in an empty room. The flow recirculations and thermal buoyancy enhanced velocity magnitude and turbulence level in the domain. Furthermore the results were influenced by turbulence model, solver and boundary conditions. The comparison with available data showed ANSYS Fluent RNG $k - \varepsilon$ turbulence model with temperature boundary condition on the heated internal box calculated the best results. Considering flow fluctuation in the 3-D benchmark cases, there is no direct relation in an empty room (case 3.2.1) between velocity and turbulent flow in power spectral density and frequency, and each of time-history velocity oscillations is independent and random. At the mid-height of the domain (Y = 0.5), the energy of the velocity fluctuation is relatively weak. Adding a unheated box in the centre of the domain (case 3.2.2) obtained larger frequency than that of case 3.2.1 and its energy level was weaker. The effect of a non-heated box in the domain on flow feature was seen on irregular oscillations above the box, caused by the clockwise recirculations above the top of the box and between the rear

62

box surface and domain back wall and faster oscillation rate at the monitoring locations. The frequency of velocity oscillation was decreased compared with that of case 3.2.2 and its values were consistent with temperature at the location although the energy of the fluctuation is much higher in temperature. The strong energy of the oscillations is found at the lowest level (i.e. Y = 0.1) at P1, P6 and P10 for velocity and through the domain height at P6 and P10 (i.e. spanwise mid-plane) for temperature, confirming that the dominant frequency depended on the orientation of the flow circulation. It can be concluded that the formation of thermal plume from the heated box stabilised flow in the upper part and the sides of the heated box on a spanwise plane.

Chapter 4

Conjugate Heat Transfer (CHT) on Indoor Thermal Comfort

This chapter is to investigate the conjugate heat transfer in a 2-D non-ventilated natural convection model room with a heating source and a 3-D ventilated forced convection model room with a heating source and window glazing. Details of flow and heat transfer characteristics will be carried out with parameters including the location of the heating source, the wall thickness and the wall thermal conductivity effects on indoor thermal condition such as comfort temperature as well as energy consumption.

4.1. Conjugate heat transfer in two-dimensional closed model

The validity of numerical conjugate heat transfer models has been assessed for a 2-D model problem including streamlines (Ψ) and isotherms (θ) at different Grashof numbers and corresponding stream function and temperature distributions at different cross-sections of the domain. The employed mathematical models and numerical schemes will be carefully tested and compared with other already validated numerical predictions and theoretical calculations (Kuznetsov and Sheremet, 2011).

4.1.1. General description

A two-dimensional model room with conjugate natural convection heat transfer has been chosen for validation against available numerical results obtained by Kuznetsov and Sheremet (Kuznetsov and Sheremet, 2011) using finite-difference CFD approach. It is a square closed model room (i.e. no ventilation, thus natural convection only) with uniform finite-thickness bounding walls and a localised heat source (see Fig. 4.1). The heat source is similar to a radiator panel which has a constant uniform volumetric thermal-power density throughout the computation. It is located at the internal surface of the left-side wall. The external surface of this wall (x = 0) is directly exposed to external environment. Other three walls $(x = x_{max}, y = y_{min} \text{ and } y_{max})$ are assumed to be adiabatic without radiation heat exchange (e.g. $\frac{\partial \theta}{\partial n} = 0$ where *n* is dimensionless distance acting normal to the surface in *x* or *y* direction). At the solid-fluid interface, the velocity components are set to be zero (i.e. no-slip condition). Because of a low Grashof number, heat source and domain size, the fluid medium inside the flow domain is incompressible air with laminar viscous flow status. The thermophysical properties of solid walls and fluid (air) are assumed constant.



Figure 4.1. Schematic view of a 2-D model room.

The model room has dimensions of H/L = 1, d/L = 0.06, $l_B/L = 0.18$, $h_B/L = 0.3$, $h_A/L = 0.18$. Two natural convection scenarios are considered at the following conditions; i.e. Prandtl number Pr = 0.7, heat conductivity ratio $k_{gas}/k_{wall} = 3.7 \times 10^{-2}$ and Biot number Bi = 2.86; and scenario 1 has Grashof number $Gr = 1.6 \times 10^7$, dimensionless environment temperature $\theta_e = -0.46$ and scenario 2 has $Gr = 2.4 \times 10^7$, $\theta_e = -0.31$. That is equivalent to a wall thermal conductivity of 0.89 W/mK and an external environment temperature of 292.8 K at two given Gr numbers while an initial temperature is set to be 293 K. Hence, a constant thermal-power density of 12.0 W/m^3 is used for scenario 1 and 17.5 W/m^3 for scenario 2, respectively along with a thermal conductivity of fluid 0.025 W/mK. The emissivity of all wall surfaces is set to be unity.

The length in horizontal (streamwise x) and vertical (wall normal y) directions are denoted as L and H, respectively and non-dimensional coordinates are X = x/L and Y = y/H.

The grid convergence study has been carried out on three successive meshes of 200×200 , 250×250 and 300×300 grid points from coarse to fine and numerical results in terms of temperature and velocity profiles (not shown) have shown no noticeable differences among three meshes, indicating results can be considered grid-independent. Thus, only results from the 200×200 mesh with non-uniformly distributed grid points are presented thereafter. The results will be compared with already validated numerical predictions (Kuznetsov and Sheremet, 2011) in terms of stream function and temperature (see section 2.9.2) at two non-dimensional locations Y = 0.35 and Y = 0.80 and at an instantaneous dimensionless time of $\tau = 500$, defined using Eq. (27).

4.1.2. Validation

Figure 4.2 shows streamlines and isotherms from two Grashof numbers at an instantaneous dimensionless time of $\tau = 500$. It can be seen that there are two large-scale circulations moving in opposite directions (i.e. counter-rotating) from two cases studied. At $Gr = 1.6 \times 10^7$, two similar size circulations are located almost horizontally, with an anti-clockwise circulation lying in the upper part of the domain with positive Ψ value, and a clockwise circulation lying in the lower part of the domain with negative Ψ value, respectively. As Grashof number increases to $Gr = 2.4 \times 10^7$, the clockwise circulation in the lower part of the domain expanded in size in the vertical direction by compressing the anti-clockwise circulation, resulting in a smaller-size anti-clockwise circulation occurred in the upper-left corner region above the heat source. For $Gr = 1.6 \times 10^7$, the strength of the anti-clockwise circulation in the upper part of the domain is about 33 % higher than that of the clockwise circulation (i.e. maximum absolute value of stream function $|\Psi|$ at the core of the circulation). On the other hand, the circulation intensities in $Gr = 2.4 \times 10^7$ differ only by 4 % between the two circulations, despite the flow circulation size is almost halved for the anti-clockwise circulation. The reason for this is probably due to cooling of the upper part of the left-side wall at $Gr = 1.6 \times 10^7$, which affects the anti-clockwise part of the convective flow. The normalised temperature field

(e.g. isotherms) shown in Figure 4.2(c) and (d) indicates that as Grashof number increases, there is a sign of thermal plume formation in the fluid domain which is stabilized in the upper part of the domain thus causing the increase of conductive heat transfer. This can be seen by the increase of temperature gradient inside the solid wall domain next to the heat source, which may lead to possible unstable stratification effect in the vertical direction above the heat source in the fluid domain.



Figure 4.2. Streamlines (Ψ) and isotherms (θ) at $\tau = 500$ for (a, c) $Gr = 1.6 \times 10^7$ and (b, d) $Gr = 2.4 \times 10^7$.

Figure 4.3 shows present results at two vertical locations Y = 0.35 and Y = 0.80, compared with published data (Kuznetsov and Sheremet, 2011). It is clear that good agreements have been achieved between present computation and previous numerical results (Kuznetsov and Sheremet, 2011) in terms of variation shape, pattern and peak locations. At a high Grashof number $Gr = 2.4 \times 10^7$, there is only one peak in the stream function magnitude ($|\Psi|$) at a low position of Y = 0.35 (see Figure 4.3(a)), but the shape and distribution of stream function have changed to a near sine wave pattern at a high position of Y = 0.8, see Figure 4.3(b). Comparing to previous predictions by Kuznetsov and Sheremet (Kuznetsov and Sheremet, 2011), present results slightly overpredicts the stream function value in a region X = [0.3, 0.6] at a low position of Y = 0.35. The temperature fields in Figure 4.3(c) and (d) show that temperature rises in both fluid and solid domains as Grashof number increases, due to the increase of the volumetric thermal-power density of the heat source (q_0) . This will result in an average temperature increase by 36.3 % on the left-side wall and 45.1 % on the top wall, between two Gr numbers tested. In the solid region X = [0, 0.06], the unstable stratification aforementioned can be seen for $Gr = 2.4 \times 10^7$ at Y = 0.8 and for $Gr = 1.6 \times 10^7$, the predicted temperature is lower than that of previous numerical results (Kuznetsov and Sheremet, 2011) at both locations of Y = 0.35, 0.8. At Y = 0.35, there is a rapid temperature drop adjacent to the heat source which may be due to fast temperature decay when moving away from the radiator. The presence of thermal plume at $Gr = 2.4 \times 10^7$ is also evident by the temperature increase in a region of 0.4 < X < 1.0 shown in Figure 4.3(d).



Figure 4.3. Comparisons of stream function and non-dimensional temperature profiles at two Gr numbers, two vertical locations of Y = 0.35 (a, c) and Y = 0.8 (b, d) and $\tau = 500$.

4.1.3. Summary of two-dimensional CHT study

Based on the conjugate natural convection heat transfer study for $Gr = 1.6 \times 10^7$ and 2.4×10^7 in a two-dimensional model room, it can be concluded that the increase of Grashof number would lead to the formation of a thermal plume, causing more conductive heat transfer. Furthermore, this would result in temperature rise in both the solid and the fluid domains and flow-intensification at the core of the circulation

magnitude (i.e. $|\Psi_{max}|$) strengthened by about 33 % for clockwise circulation and only by about 5 % for anti-clockwise circulation, respectively.

4.2. Conjugate heat transfer in three-dimensional model

Based on the validation of a 2-D model room, a 3-D ventilated model room configuration with heating source and window glazing was next studied and validated.

4.2.1. General description

4.2.1.1 Non-CHT model room

The configuration considered here is a 3-D model room previously studied experimentally by Olesen et al. (Olesen et al, 1980) and also numerically by Myhren and Holmberg (Myhren and Holmberg, 2009), see Figure 4.4. Although the experiment used finite-thickness solid walls, it was not considered in the numerical study carried out by Myhren and Holmberg (Myhren and Holmberg, 2009). This configuration includes a double panel radiator as heat source, a window, and a ventilation system (i.e. inlet above the window for extracting cold fresh air, and outlet on opposite wall for exhausting warm air), respectively and the model room has dimensions of L = 4.8 m, H = 2.6 m, W = 2.4 m, resulting aspect ratios of H/L = 0.54, W/L = 0.50. The dimensions and the location of ventilation system, radiator and window glazing can be seen in Table 4.1. The window wall with inlet duct is directly exposed to the outside environment. The origin of the coordinate system is located at the mid-point of the intersection line between the floor and the inner wall surfaces along the spanwise direction, same as that used by Myhren and Holmberg (Myhren and Holmberg, 2009).

	Size	Position in room
Inlet	Height $h_{3b}/H = 8 \times 10^{-3}$, Width $W_{inlet}/W = 0.21$	Above window at $h_{3a}/H = 0.04$, $h_{3c}/H = 0.18$ from ceiling
Outlet	Height $h_5/H = 0.02$, Width $W_{outlet}/W = 0.33$	On opposite wall, $h_4/H = 0.06$ down from ceiling
Window	Height $h_2/H = 0.46$, Width $W_{window}/W = 1.0$	Above radiator, $h_1/H = 0.31$ from floor
Radiator	Height $h_B/H = 0.23$, Width $W_{radiator}/W = 0.58$, Thickness $L_{radiator}/L = 0.01$, Panel gap $L_{panel-gap}/L = 0.008$	Underneath the window, $h_A/H = 0.02$ above floor and $L_{gap}/L = 0.02$ from adjacent wall

Table 4.1. Specification of a 3-D model room

Table 4.2 lists boundary conditions for non-CHT model as those used by Myhren and Holmberg (Myhren and Holmberg, 2009). Present simulation also uses same thermophysical properties of the fluid (air) as that of study (Myhren and Holmberg, 2009). Due to very low speed of incoming cold airflow, incompressible flow assumption is used with Pr = 0.7. Based on physical condition of the heat source, i.e. $Gr/Re^2 \gg 1$, and $Re_{inlet} > 700$, the heat transfer due to natural convection will play a major role in the heat transfer process, compared to forced convection mode. The corresponding Rayleigh number (Ra) is $Ra = 10^8$. The initial indoor temperature is set to be 16 °C based on an ambient room condition. It assumes that the window wall is a single-layer solid wall with a total *U*-value (i.e. overall heat transfer coefficient) of 0.3 W/m^2K regardless external conditions such as temperature. The window surface also has a fixed temperature of 14 °C. At the outlet, the flow is assumed to be no pressure gradient and no impact on the inlet flow in mass balance as mentioned in section 3.1.1.

Table 4.2. Boundary conditions of non-CHT model

Inlet	Uniform & constant $T_{air} = -5$ °C and $U_{air} = 0.7 m/s$
Outlet	Naturally outflow
Window	Uniform & constant temperature $T_{window} = 14 ^{\circ}\text{C}$
Radiator	Uniform & constant temperature $T_{radiator} = 42 \ ^{\circ}\text{C}$
Walls	Wall exposed to external environment: U -value = 0.3 $W/m^2 K$, Other walls: adiabatic

4.2.1.2 CHT model room with finite-thickness wall

In order to consider the effect of finite-thickness wall used in the experiment, a conjugate heat transfer configuration with a single-layer solid wall structure of width d/L = 0.063, is introduced for the window wall that is directly exposed to the external environment. Other walls are still treated as infinitely small thickness, same as the study of Myhren and Holmberg (Myhren and Holmberg, 2009). The boundary conditions for the finite-thickness wall are applied with the following assumptions of external environment: the outer surface of the solid wall has the same temperature as external environment (-5 °C) and heat transfer coefficient $h_{out} = 34.0 W/m^2 K$ commonly used as the Winter season condition for industrial applications. The external surface of the window also assumes to be the same temperature of the external environment and the window thermal conductivity is defined as $k_{air}/k_{window} = 0.03$, which gives k_{window} about 0.9 W/mK (Cengel and Ghajar, 2011). The radiation heat exchange is only considered for the finite-thickness window wall.



Figure 4.4. Schematic view of 3-D CHT configuration.

In order to compare present results with available thermal comfort data obtained by previous experimental and numerical studies (Myhren and Holmberg, 2009) (Olesen et al, 1980), four monitoring lines are inserted at locations of x = 0.6 m, x = 1.8 m, x = 3.6 m and x = 4.2 m from the coordinate origin on a streamwise mid-plane throughout the domain height, i.e. P1, P2, P3, P4 lines, respectively and with a reference line positioned at the centre of the computational domain.

Due to a high Richardson number $Ri = Ra/(Re^2 Pr) > 1.4 \times 10^4$, numerical instabilities in terms of oscillations in convergence history and flow patterns occurred during the steady RANS computation, and the phenomenon is similar to that observed by Raji et al. (Raji et al, 2008). Note that *Ra* is Rayleigh number and *Re* is Reynolds number. This is partly due to the reason that there may exist moderate to strong thermal instabilities caused by the presence of a heat source in the lower region and a ventilation cold airflow in the upper region of the wall, resulting in the formation of a thermal plume and heat exchange between cold and warm airflows inside the domain (Antonelli et al, 2003). Hence, temperature and velocity results are averaged using three sets of time history data at the monitoring lines P1-P4, with maximum temperature and velocity variations kept within 1 °C and 0.05 m/s range, respectively. The average results are then used to compute the heat transfer of the radiator panels and the room comfort temperature by using Eq. (36), for the monitoring points. Results are then compared with available validated data from other numerical studies (Myhren and Holmberg, 2009). A careful grid convergence study was performed using a block-structured mesh, and the mesh with grid points in a range of 110,000 to 130,000 is finally generated for all test cases presented here.

4.2.2. Validation

Numerical results from present study compared with those from commercial numerical code, FloVENT (Myhren and Holmberg, 2009) in terms of fluid temperature and radiator surface heat transfer are shown in Table 4.3 and with theoretical estimation in terms of solid wall surface temperature and heat transfer shown in Table 4.4. Note that heat transfer from heat source is computed using formula $Q = Q_{convection} + Q_{radiation}$, and that theoretical values in Table 4.4 are calculated using Eqs. (32) and (33) based on numerically calculated heat flux and temperature with the assumption of $T_{out} = -5$ °C, $U = 0.3 W/m^2K$ and 22 °C of target ambient indoor temperature, respectively. Also the average of surface temperature is computed using a formula $\bar{\phi} = \frac{1}{A} \int \phi \, dA$ over a control volume (where ϕ is integration variable).

It can be seen from Table 4.3 that the differences between present prediction and those from previous studies are very small in terms of dimensionless temperatures and heat transfer coefficients from the radiator. In general, the non-CHT model predicts temperature slightly lower than that of the CHT model. The present results also show slightly a lower total heat transfer but a significantly higher radiative heat transfer, compared to that obtained by Myhren and Holmberg (Myhren and Holmberg, 2009). There is no noticeable difference between CHT and non-CHT results. Comparison between present prediction and theoretical estimation shows that the predicted bottom wall temperature is higher than that of theoretical value, probably due to the existence of heat source next to the wall, and this may result in the higher corresponding wall heat flux and heat transfer of wall, as seen in Table 4.4. Figure 4.5 gives the comfort temperature distributions at monitoring lines (P1 - P4) and reasonably good agreements between three sets of predicted values have been achieved in terms of shape variation, pattern and peak locations, with temperature differences within a small range of ± 0.5 °C. The influence of CHT is small near bottom wall region and becomes slightly larger near upper wall region.

	FloVENT	Non-CHT model	CHT model
T _{air} at 1.1 m level at ref line (°C)	20.8	21.0	21.6
T _{comfort} at 1.1 m level at ref line (°C)	21.0	21.2	21.8
Average Q _{total} of radiator (W)	483	461	461
Average Q _{radiation} of radiator (W)	180	215	221
Average HTC of radiator (W/m ² K)	6.7	6.6	6.7

Table 4.3. Comparison of fluid temperature and heat transfer from the radiator

Table 4.4. Comparison of surface temperature and heat transfer of the wall

	Theory	Non-CHT model	CHT model
Mean T_{inner} of top surface of wall (°C)	21.0	21.0	20.4
Mean T_{inner} of bottom surface of wall (°C)	21.0 ^ª	25.6	27.6
Mean T_{inner} of window surface (°C)	14.0	14.0	14.6
Heat flux through wall (W/m^2)	8.10	13.54	10.52
Overall HTC through wall (W/m^2K)	0.31	0.49	0.37

^a based on assumption in section 4.2.2.

. Х.1



Figure 4.5. Comparison of comfort temperature profile at monitoring lines with FloVENT.

Figure 4.6 shows comparison of fluid temperature distributions from CHT and non-CHT model cases on the solid-fluid interface (x = 0 m) at two spanwise locations of z = 0 m and z = 0.9 m. The results show that there are clear differences in temperature distributions inside the solid wall domain between two models. At z = 0 m of midspanwise location, CHT model predicts higher temperature mainly in the bottom wall region where the heated air from the radiator panels moves upwards and meets the cold air from the inlet where a large temperature drop is observed at y = 2.1 m location. Overall, maximum difference between CHT and non-CHT models of about 10 % occurred in a region of y(m) = [0, 0.8] in the vicinity of the heat source, and this difference reduces to about 6 % in a region of y(m) = [2.0, 2.6] near the inlet opening location and the top wall. In the CHT model results, a non-uniform temperature distribution is observed along the window height, while non-CHT gives uniform distribution as expected. At z = 0.9 m location near side wall, the differences between two models are very small, as the position is away from the heating panels and the inlet opening.



Figure 4.6. Comparison of temperature variation at x = 0 m at (a) z = 0 m and (b) z = 0.9 m.

Figure 4.7(a) shows the non-dimensional temperature distributions in mid-plane (x, y, z = 0) throughout the solid wall domain and partly fluid domain containing the radiator, using the CHT model. It can be seen that the bottom part of the solid wall has been heated up by the nearby heat source (radiator), whereas cooling loads are persistent near outer wall region (due to cold environment temperature used as boundary condition) and near the inlet slot. Figure 4.7(b) gives non-dimensional temperature distributions inside the solid domains at three heights $(0.5h_1, 0.5h_2 \text{ and } 0.5h_3)$ as seen in Figure 4.4 in the vertical direction on a streamwise mid-plane, compared with theoretical estimation based on one-dimensional heat conduction Eq. (6), i.e. T = 86.2 x + 21.0 for $0.5h_1$ and $0.5h_3$ positions and T = 69.8 x + 14.0 for $0.5h_2$ position. It can be seen that there are good agreements between numerical predictions and theory at $0.5h_2$ and $0.5h_3$, corresponding to the window and the upper part of the solid wall. However, there is clear disagreement at $0.5h_1$, probably due to the strong influence of the heat source on the bottom part of the solid wall, where theoretical calculation based on 1-D heat conduction equation does not consider the effect of heat source. The CHT model prediction showed that there is considerable increase in the amount of heat passing through the wall surface at the $0.5h_1$ about 11.12 W/m^2 of wall heat flux. Therefore it can be concluded that the predicted temperature from the CHT model with finite-thickness wall is only sensitive in

the area close to the heating source and the effect reduces rapidly while away from the source.



Figure 4.7. Distributions of (a) isotherms ($T \circ C$) in the solid wall and (b) temperature distributions at three vertical heights compared to theoretical estimation at a streamwise mid-plane by the CHT model.

4.3. Parametric study in a 3-D CHT model room

Building on previous success of validation and verification exercises of several benchmark test cases including natural convection in a tall cavity (Betts and Bokhari, 2000) and 2-D and 3-D conjugate heat transfer models using a commercial CFD code ANSYS Fluent (Horikiri et al, 2014), CHT model results demonstrated its suitability to simulate the flow and heat transfer in an indoor environment. Present study performs parametric analysis for better thermal comfort using the 3-D conjugate heat transfer model, i.e. a 3-D ventilated forced convection model room with a heating source and window glazing. Details of flow and heat transfer characteristics will be carried out with parameters including the location of the heating source, the wall thickness and the wall thermal conductivity effects on indoor thermal condition such as comfort temperature as well as energy consumption. The employed mathematical models and numerical schemes

will be carefully tested and compared with other already validated numerical predictions (Myhren and Holmberg, 2009) and theoretical calculations (Cengel and Ghajar, 2011).

Design optimisation aims to achieve better indoor thermal comfort, and a study has been conducted by a wide range of parameter studies, such as the arrangement of heat source and window glazing based on the CHT model room (a total of six cases), wall thickness variations (a total of two cases), and the wall material property of thermal conductivity sensitivity (a total of four cases), respectively (see Table 4.5). These parameters were chosen as close as possible to realistic domestic room conditions. For example, wall thickness of d/H = 0.042 and 0.083 were considered, based on the minimum exterior wall thickness of detached houses and flats in the UK to have an average U-value of 0.22 $W/m^2 K$ (Department for Communities and Local Government, 2007). The wall thermal conductivity is taken from concrete, bricks and well-insulated walls, e.g. the average U-value for UK residential properties with a wall thickness of 0.3 m, $k_{air}/k_{wall} = 0.12$. Other geometry and boundary conditions used in above 3-D computation remained the same as the original configuration. Due to geometric constraints, in particular the heights of radiator panel $h_B/H = 0.31$ and window glazing $h_2/H = 0.46$, heat source is re-located at a position of $h_A/H = 0$ on the floor wall. otherwise it remains at $h_A/H = 0.02$ above the floor level. Also the location of window is arranged at $h_1/H = 0.38$ from the floor wall for the case of window size of $h_2/H =$ 0.38. The results will be compared with previously validated CHT mode predictions (i.e. $h_B/H = 0.23$ and $h_2/H = 0.46$) in terms of comfort temperature and heat loss magnitude at monitoring points and on wall surfaces etc.

Case (size arrange	4.3.1 ment study)	Case 4.3.2 (wall thickness study)	Case 4.3.3 (wall material study)
Window	Radiator	Wall thickness	Thermal conductivity
h_2/H	h _B /H	d/L	k _{air} /k _{wall}
0.38	0.15	0.042	0.02
0.46	0.23	0.083	0.04
	0.31		0.08
			0.12

Table 4.5. Parametric case studies

4.3.1. Effects of heat source and window glazing sizes (Case 4.3.1)

Figure 4.8 gives comparison of heat transfer and corresponding volume-averaged comfort temperature for various heights of heat source (h_B/H) and window glazing (h_2/H) . The heat transfer Q(W) is calculated based on numerically calculated heat flux and area of the wall. The volume-averaged comfort temperature is computed using the formula $\frac{1}{v} \int \phi dV = \frac{1}{v} \sum_{i=1}^{n} \phi_i |V_i|$ where V is volume. It can be seen that while the size of h_B increases, the comfort temperature increases accordingly.

For a given radiator panel size (h_B/H) , the comfort temperature is lowered by about 5 % for large window glazing (h_2/H) (i.e. about 20 % increase in window surfacearea), due to increased heat loss through the glazing. Among three different radiator sizes, the volume-averaged comfort temperature differs by a maximum of 5.4 °C in case of small window glazing $(h_2/H = 0.38)$ and 6.3 °C in case of large window glazing $(h_2/H = 0.46)$. It is clear that using a small radiator of $h_B/H = 0.15$ with a large window glazing $(h_2/H = 0.46)$, it is difficult to sufficiently heat the entire domain, whilst it can be slightly overheated by using a large radiator $h_B/H = 0.31$ with a small window glazing. Note that the international standards recommend the comfort temperature to be between 20 °C and 24 °C (International Organization for Standardization, 2007). For a small-size heat source, the buoyant warm air may be too weak to heat the cold window-surface and to 'block' the cold inlet flow downward, as a result of the location of the heat source, i.e. too close to the floor. In contrast, a large-size heat source located just below the window can block the development of a cold zone near the inlet, and sometimes it may even lead to overheating. Despite thermal temperature difference at the given radiator panel size (h_B/H) , energy consumption through the radiator panels calculated using the same method described in section 4.2.2. is comparable for small and medium sized radiators $h_B/H = 0.15$ and 0.23, indicating that a large window glazing lets out more energy from the room domain. In the case of a large-size radiator $h_B/H = 0.31$, energy loss increases by 6.4 % as the window glazing size (h_2/H) increases. There is only a small increase in heat transfer for a large-size radiator of $h_B/H = 0.31$ with a small window glazing size $h_2/H = 0.38$, compared with a medium-size radiator $h_B/H = 0.23$. In the case of a medium radiator panel of $h_B/H =$ 0.23, the estimated energy consumption by the heat source is about 460 W, while a small size radiator panel of $h_B/H = 0.31$ consumes 22 % less energy and a large size radiator.



Figure 4.8. Variations of heat transfer through the radiator as heat source (two bottom lines) and the corresponding comfort temperature (two top lines) for three different sizes of radiator (h_B/H) and window glazing (h_2/H) .

4.3.2. Effect of wall thickness (Case 4.3.2)

For a fixed radiator panel of $h_B/H = 0.23$ and a window glazing $h_2/H = 0.46$ arrangement, the effect of wall thickness is studied with either a thicker or thinner solid wall in comparison with the original wall thickness (i.e. d/L = 0.063), applying CHT model. The resultant comfort temperature and heat transfer were derived for comparison with the original wall thickness from the present study.

Figure 4.9 shows heat transfer rate at the solid-fluid interface (x = 0 m) of the solid wall (excluding the window glazing part) and the corresponding volume-averaged comfort temperature at given wall thickness. It can be seen that the heat transfer decreases as the wall thickness d/L increases, indicating that heat loss through the solid wall would be reduced with the increase of wall thickness d/L although the difference between heat transfer at d/L = 0.063 and 0.082 is very small. Furthermore, there is more heat loss seen from the bottom section of the wall (i.e. h_1) due to the location of heat source. Overall, the heat loss through the bottom section of the wall is higher by about 9 – 22 % than the upper part of the wall (i.e. h_3) and this contributes towards about 55 - 61 % of the total heat loss from the solid wall. Compared with the original wall thickness d/L = 0.063, total heat loss through the solid wall could be increased by 35 % for a thinner wall d/L = 0.042 but decreased by 12 % for a thicker wall d/L =0.082, respectively. The corresponding heat loss through the bottom section of the wall increases by 42% for a thinner wall and decreases by 17% for a thicker wall, respectively. The comfort temperature is also significantly influenced by the wall thickness, resulting in an average difference value of $\Delta T = 4.6$ °C between thinner wall d/L = 0.042 and thicker wall d/L = 0.083. The comfort temperature difference in comparison to the original wall thickness is about 9 % decrease in case of a thinner wall and 11 % increase in case of a thicker wall. Overall, the comfort temperatures with thinner and original wall thickness of d/L = 0.042 and 0.063 both satisfy the nominal building requirements, while domains with a thicker wall with d/L = 0.083 will be slightly overheated, possibly leading to energy wastage unless radiator heating temperature is set to be at a lower level. As a result, wall thickness d/L = 0.063 would be sufficient for indoor thermal comfort while keeping an acceptable level of heat loss, while there is a glazed window and an air inlet.



Figure 4.9. Variations of heat transfer (solid lines) at solid wall surfaces and the corresponding comfort temperature (dashed line) at different wall thickness as d/L.

4.3.3. Effect of wall thermal conductivity (Case 4.3.3)

Numerical studies of wall thermal conductivity effect were carried out at fixed radiator panel height, window glazing height and wall thickness of $h_B/H = 0.23$, $h_2/H = 0.46$ and d/L = 0.063, respectively.

Figure 4.10 gives the heat transfer rate at the solid-fluid interface (X = 0) of the wall and volume-averaged comfort temperature with various thermal conductivity of the solid wall. It can be seen that the comfort temperature decreases at lower thermal conductivity ratios. The difference in comfort temperature between $k_{air}/k_{wall} = 0.02$ and 0.28 is about 4 °C while the difference between $k_{air}/k_{wall} = 0.12$ and 0.28 is reduced significantly to about 0.4 °C. It is expected that the heat loss would be increased significantly at lower thermal conductivity ratios (i.e. k_{air}/k_{wall}). For example, the heat loss could be increased by 8 times in the bottom section of the solid wall and by 6 times in the top section of the solid wall, respectively for $k_{air}/k_{wall} = 0.02$ and 0.28 (see Figure 4.10). Despite the walls with $k_{air}/k_{wall} = 0.08$ and 0.12 both representing wellinsulated walls, there will be 2 - 3 times more heat loss compared with that of the original ratio of $k_{air}/k_{wall} = 0.28$.



Figure 4.10. Variations of heat transfer (solid lines) at solid wall surfaces and the corresponding comfort temperature (dashed line) at different wall thermal conductivity ratios k_{air}/k_{wall} .

4.4. Summary

A systematic investigation of conjugate natural convection heat transfer in a ventilated room with localised heat source and window glazing has been carried out by using a computational fluid dynamics approach, with results carefully validated against published data in literature (Myhren and Holmberg, 2009), (Olesen et al, 1980), (Cengel and Ghajar, 2011). After validation, the model has been used to investigate the effects of heat source and window glazing arrangements, wall thickness and material property variation on indoor thermal performance. The results showed that the heat source and glazing sizes have significant impact on temperature field, and the wall thickness and thermal conductivity also revealed considerable impact on the level of energy consumption through the solid wall (i.e. heat loss), especially from the bottom section of the wall. The heat energy loss through the solid wall surfaces (i.e. adjacent to the radiator panels) was about 35 % when reducing the wall thickness by 10 *cm* from the original wall thickness 30 cm to 20 cm. This would reduce thermal comfort level of the domain. In fact, the

volume-averaged thermal comfort was decreased by 9% compared with that when original wall thickness was used. The large amount of heat loss is mainly influenced by the heat source being next to the solid wall without suitable insulation. With the minimum wall thickness to meet UK's domestic house requirements, the thermal comfort can be sustained within the indoor environment standards. However, the total heat loss through a thinner wall of 20 cm thickness is about 53% high, compared with that through a thicker wall of 40 cm thickness. In a model room configuration as studied here, ideal indoor thermal environment can be achievable with a radiator size of $h_B/H = 0.23 - 0.31$, window glazing size of $h_2/H = 0.38 - 0.46$, wall thickness of d/L = 0.042 - 0.063, and thermal conductivity ratios of $k_{air}/k_{wall} = 0.28$ can be applied to the region that has warmer Winter conditions. However for cold Winter conditions, a large size radiator panel, well-insulated walls, and a low wall thermal conductivity are required.

Chapter 5

Thermal Comfort and Heat Transfer in Furnished and Occupied Room

This chapter is to investigate indoor thermal comfort in three-dimensional model room that is used in Chapter 4 (the conjugate heat transfer study) but without wall thickness on a window wall (see Figure 5.1(a)). The aim of study is to have better understanding of heat transfer in furnished and occupied room and hence to improve indoor thermal comfort of the occupants. After the careful validation on a 3-D empty model room against published data in literature, the model has been used to investigate the effect of furniture arrangement with and without heat generation and occupants on indoor thermal comfort. The results of furnished and occupied room heat transfer is used to analyse thermal comfort around occupant in the domain and hence to improve indoor thermal comfort of the occupants using Fanger's indices (e.g. PMV-PPD indices) in an in-house FORTRAN code (International Organization for Standardization, 2007), under the specific conditions of occupant and indoor environment.

5.1. Heat transfer and thermal condition in furnished and occupied room

5.1.1. General description

Analysis of the impact of occupied room on indoor thermal comfort is carried out by three different layouts/scenarios with furniture and/or occupants (S_1, S_2, S_3) (see Figure 5.1(b-d)), compared with the original empty model room layout/scenario S_0 in Figure 5.1(a). The detailed descriptions of installation location for the original empty model room can be found in Table 4.1.

The furniture considered a cabinet (or a TV stand) with a TV at a fixed position, located at the middle of one side-wall opposite to the sofa, and two different types of sofa. A small sofa that has no armrest is located at the back wall, facing to the window wall (denoted as the layout S_1) while a large sofa with armrest is located at the middle of one side-wall (denoted as the layout S_2). In the layout S_3 , two sofas are both included. All sofas and cabinet/TV-stand are attached to the walls, assuming that the gap between the walls and non-heat generating furniture is so small that the local heat transfer and fluid pattern inside the gap space do not have significant influences on the domain of interest. A total of three different room layouts (S_1 , S_2 and S_3 , see Figure 5.1) are studied with max three different heat transfer modes by introducing corresponding energy sources (e.g. radiator, occupant(s) and TV) that are designed to incrementally increase the complexity of geometry features.

Table 5.1 shows the presence of heat source in each case; i.e. Case 5.1.1 has only one radiator in furnished room, Case 5.1.2 has occupant(s) relaxing on a sofa without TV and Case 5.1.3 has occupant seated on a sofa, watching a TV.

Table 5.1. Case study with heat generation source

Case	Radiator	Occupant(s)	TV
5.1.1 (radiator study)		×	×
5.1.2 (thermal human study)	\checkmark	\checkmark	×
5.1.3 (heat-generating TV study)	\checkmark	\checkmark	\checkmark

For further studies of heat generation, a box-shaped human being is introduced at the center of each sofa. An occupant with a small-shoulder (H_1) is seated on the small sofa S_1 , facing to the window wall, while another occupant with a large-shoulder (H_2) is seated on the large sofa S_2 , facing to the cabinet/TV-stand on the opposite side-wall. The bodies are seated along the sofa without gap/space and therefore the total height from the feet to the head is 1.3 m (i.e. 0.3 m as the height of the head, 0.6 m as the upper-body from the shoulder to the seat, and 0.55 m from feet to knees with 0.15 m leg thickness, respectively), see Figure 5.2. The details of dimensions and location of furniture and human being can be found in Table 5.2.



Figure 5.1. Schematic views of four 3-D configurations of furniture with monitoring four lines (P1 - P4): (a) layout S_0 , (b) layout S_1 , (c) layout S_2 , (d) layout S_3 .





During data analysis such as PMV and PPD values of thermal comfort conditions for occupant in a relaxing mode on the sofas, the area surrounding the seated occupant is considered as a cylindrical shape with a 0.5 *m* radius of circle from the centre of seated human beings, from the floor to the 1.3 *m* level (i.e. the top of the head), so-called "occupied zone", see Figure 5.3. The measurements are taken at eight points across the occupant's body (see Figure 5.3(b)) at four vertical levels from the floor to the ceiling, i.e. Y = 0.1 m (ankle level), 0.5 *m* (knee level), 1.1 *m* (shoulder level) and 1.3 *m* (head level), respectively.



Figure 5.3. Detail of seated occupant's segments (a) side-view, and occupied zone with eight measuring points (b) top-view.

	Size $(L \times H \times W)$ (m^3)	Position in room
Sofa S ₁	Outline: $1.4 \times 0.8 \times 0.8$, Seat: $1.4 \times 0.4 \times 0.6$	Mid-position along the back-wall
Sofa S ₂	Outline: $1.9 \times 0.8 \times 0.8$, Arms: $0.8 \times 0.25 \times 0.25$, Seat: $1.4 \times 0.4 \times 0.6$	Mid-position along the side-wall
Cabinet	$1.9 \times 0.8 \times 0.8$	Mid-position along opposite side- wall of the sofa
TV	$0.9 \times 0.6 \times 0.5$	Centre/top of cabinet
Human H ₁	Body: 0.5 × 0.6 × 0.3, Head: 0.2 × 0.3 × 0.3, Thigh & leg:0.5 × 0.55 × 0.15	Seating at centre of sofa S_1 /no gap to sofa surface
Human H ₂	Body: $0.6 \times 0.6 \times 0.3$, Head: $0.2 \times 0.3 \times 0.3$, Thigh & leg: $0.6 \times 0.55 \times 0.15$	Seating at centre of sofa S_2 /no gap to sofa surface

Table 5.2. Specifications of furniture and human body

Table 5.3 lists boundary conditions for the baseline case S_0 , same as those previously defined and used (Myhren and Holmberg, 2009) and also for other three cases in the present study (i.e. layouts S_1 , S_2 and S_3). The plastic-made TV has a thermal conductivity value of 0.2 W/m. K and it generates a constant heat of 2000 W/m^3 for "on-mode" and no heat for "off-mode", respectively. A 1.30 *m*-height human being in seated condition has a mean surface temperature of a human body of 31 °C in a relax mode, equivalent to constantly releasing 75 – 85 W heat from total body volume of 0.18 – 0.20 m^3 (Bojic et al, 2002), (Sorensen and Voigt, 2003), (Zhuang et al, 2014). The radiator panels also have a constant heat generation to maintain the volume temperature of the panel around 40 - 42 °C in each case.

Present study uses the same thermo-physical properties of the fluid (air) as that of previous study (Myhren and Holmberg, 2009). Due to very low speed of incoming cold airflow, incompressible flow assumption is used with a Prandtl number Pr = 0.7. Based on physical condition of heat source, i.e. $Gr/Re^2 \gg 1$, and $Re_{inlet} > 700$, where Gr is Grashof number, Re is Reynolds number, the heat transfer due to natural convection

mode will play a major role in the heat transfer process, compared to forced convection mode. The corresponding Rayleigh number (*Ra*) is $Ra = 10^8$. The initial indoor temperature is set to be 16 °C based on an ambient room condition.

Inlet	Uniform & constant, $T_{air} = -5$ °C and $U_{air} = 0.7 m/s$
Outlet	Naturally outflow
Window	Uniform & constant temperature, $T_{window} = 14 ^{\circ}\text{C}$
Walls	A wall exposed to external environment, U-value = $0.3 W/m^2$. K Other walls: Adiabatic
Radiator	Constant heat generation to keep $40 - 42 ^{\circ}C$
TV	Constant heat generation 2000 W/m^3 for "on-mode"
Humans	Constant body temperature 31 °C
Sofa	Non-heat generating furniture, Adiabatic
Cabinet	Non-heat generating furniture, Adiabatic

Table 5.3. Boundary conditions

The results are compared with those available thermal comfort data obtained by previous experimental (Olesen et al, 1980) and numerical studies (Myhren and Holmberg, 2009) at the aforementioned monitoring locations (P1, P2, P3, P4) on a streamwise midplane throughout the domain height, see Figure 5.1(a).

The study has been carried out by performing steady RANS computations using ANSYS Fluent software for indoor thermal comfort temperature prediction, and thermal index calculations using an in-house FORTRAN code for Fanger's PPD index evaluation (International Organization for Standardization, 2007), respectively. The results are compared in comfort temperature with available data from another commercially available numerical code FloVENT (Myhren and Holmberg, 2009), whereas the accuracy of predicted PPD magnitude has been analysed in the conditions such as the occupant relaxed on a sofa (i.e. 1.0 met) wearing the winter indoor clothes (1.0 clo) with 50% of air humidity for different sizes of single-panel radiator and ventilation system in the domain (Myhren and Holmberg, 2008). The obtained results are compared on a streamwise mid-plane and at aforementioned four monitoring lines (P1 - P4).
5.1.2. Validation

The validity of numerical thermal comfort assessment has been analysed for the same 3-D model in the previous section but without wall thickness (see section 4.2.2) using comfort temperature and Fanger's thermal index (e.g. PPD) with other published numerical data and theoretical values in the previous section along with CHT model.

Figure 5.4 shows comparison of predicted comfort temperature and PPD index profiles at four monitoring locations with available published data (Myhren and Holmberg, 2008), (Myhren and Holmberg, 2009). It is clear that reasonably good agreements between two predicted values have been achieved in terms of shape variation and pattern except the location of P1, with maximum differences within a small range of ± 0.5 °C and ± 2.5 %. The comfort temperature increases at three downstream locations P2 - P4 as the domain height increases. At location P1, the comfort temperature decrease in the upper part of the domain (0.6 m < y < 1.8 m) are possibly caused by the influence of nearby low temperature glazing window and cold jet stream. Compared with that of FloVENT, the present comfort temperature profiles have shown a slight over-prediction throughout the domain height, especially in the region below y = 1.1m in height. The PPD calculations shown in Figure 5.4 indicate a similar trend as FloVENT (Myhren and Holmberg, 2008) that thermal comfort level increases whilst towards the ceiling. Some over-predictions in present results at locations P2 - P4 may be due to larger amount of heat transfer from the double-panel heat source, causing higher temperature, radiation temperature and velocity. With the same reasons, the calculated average PPD value at each location is approximately 3.5 % higher than that of experiment data (Olesen et al, 1980). The difference in PPD distributions between two sets of prediction data is also recognisable at location P1 where there exists a strong thermal flow mixing between a cold jet stream from the inlet slot and a warm air stream from the heat source beneath it. More results validation against available test data (Myhren and Holmberg, 2008), (Olesen et al. 1980). (Zuo and Chen, 2009) can be found in a recent publication (Horikiri et al, 2014), using same mathematical model and numerical scheme.



Figure 5.4. Comparison of comfort temperature and PPD profiles at four monitoring locations (P1 - P4) between present study and published FloVENT results (Myhren and Holmberg, 2008), (Myhren and Holmberg, 2009).

5.2. Case study

The computational results of each study case are presented in terms of comfort temperature as well as velocity and air temperature, comparing with other numerical data (Myhren and Holmberg, 2009) and previous already validated numerical results in an unfurnished model room (Horikiri et al, 2014). The flow field data were used to calculate Fnager's PMV-PPD indices in an in-house FORTRAN code (International Organization for Standardization, 2007) under the conditions such as the occupant relaxed on a sofa (i.e. 1.0 met) wearing the winter indoor clothes (1.0 clo) with 50% of air humidity.

5.2.1. Effect of furniture arrangement (Case 5.1.1)

Figure 5.5 gives three-dimensional velocity magnitude distributions at iso-surface of $> 0.1 \ m/s$, with coloured with the x-velocity (u) contours in a range of $-0.1 \ m/s$ to $0.1 \ m/s$ for all four layouts ($S_0 - S_3$). It can be seen that velocity at $0.1 \ m/s$ and higher are clearly visible along the surfaces of floor, ceiling and around furniture. Due to the fact that furniture is located too close to the main stream path from the inlet, the furniture are

regarded as obstacles along the flow path, causing shear flows with high velocity gradients at the edges of the furniture and nearby regions, that further leading to recirculation flows in anti-clockwise direction in the lower space between the window wall and the furniture.



Figure 5.5. Velocity magnitude iso-surface of 0.1 m/s coloured with x-velocity (u) contours for four room layouts: (a) S_0 , (b) S_1 , (c) S_2 , (d) S_3 .

Figure 5.6 shows comfort temperature distributions at four monitoring locations P1 - P4 throughtout the domain height for three room layouts with sofa $(S_1 - S_3)$, compared with other numerical results of FloVENT (Myhren and Holmberg, 2009) and Fluent for an empty room layout of (S_0) (Horikiri et al, 2014). It is clear that the predicted comfort temperature in furnished room layouts is generally in good agreement with that of unfurnished room. Note that temperature at the location P1 where the maximum difference of comfort temperature is about 2 °C at a height of 1.5 m, is highly affected by the cold stream from the inlet and the additional re-circulations mentioned beforehand would cause some rapid changes in flow pattern and other features such as velocity hence

leading to the change in comfort temperature. There is little difference among four layouts in terms of the comfort temperature magnitude at other three monitoring locations P2 - P4, indicating that the fluid is less influenced by the existence of furniture in a streamwise plane at the centre (z = 0 m). The predicted average air temperature in the fluid domain is around 21.5 °C, calculated using a formula $\frac{1}{v} \int \phi dV = \frac{1}{v} \sum_{i=1}^{n} \phi_i |V_i|$, which is consistent among all case studied.



Figure 5.6. Comparison of comfort temperature profile at four monitoring locations (P1 - P4) from four room layouts $S_0 - S_3$ with that of FloVENT for an empty model room layout S_0 (Myhren amd Holmberg, 2009).

5.2.2. Effect of heat transfer from occupant (Case 5.1.2)

Figure 5.7 shows predicted air temperature contours at a streamwise mid-plane (z = 0 m) for three different layouts $S_1 - S_3$ with sofa. It can be seen that air temperature increases with the height and the formation of thermal plume from human bodies can be clearly observed. The trends of temperature elevation and thermal plume from human body were observed in various studies (Niu et al, 2001), (Nilsson, 2004), (Lin et al, 2005). While increasing the number of occupant from one to two (i.e. layout S_3), there is air temperature rise along the vertical height, causing a large size of thermal plume occurred in the fluid domain which is found more stable in the upper part of the flow domain. The

air temperature gradient magnitudes at two vertical levels of 0.1 m and 1.1 m from the floor are predicted 1.4 °C, 1.3 °C and 0.7 °C, respectively for three layouts S_1 , S_2 and S_3 , for which they satisfy the ISO thermal comfort standard (International Organization for Standardization, 2007). The air temperature surrounding the human being H_1 changes by maximum 2 °C between two layouts S_1 and S_3 . These results confirm that the presence of thermal occupant does have influences on indoor environment temperature, with increased volume-averaged temperature of 1.7 - 3.2 °C for three layouts $S_1 - S_3$, compared to that of unoccupied room, i.e. case 5.1.1. The corresponding change in comfort temperature rise is measured about 8.7 % - 16.3 %. Among all three different room configurations, the averaged comfort temperature is in a range of 23.3 °C - 24.9 °C, equivalent to 5.9 % - 6.5 % in difference, compared with that of the layout S_3 in which high fluid (air) temperature is predicted. It is thus concluded that the increase of the number of thermal occupant would lead to the air temperature increase of maximum 3.2 °C.

Figure 5.8 shows the distributions of velocity magnitude contours at a vertical plane through the mid-width of occupant's head, i.e. x = 4.45 m for human being H_1 in Figure 5.8(a), 6(c) and z = 0.85 m for human being H_2 in Figure 5.8(b), 6(d), illustrating the formation and development of thermal plume from each occupant's body. It is clear that the rising thermal plume is of significant strength with a maximum velocity above 0.14 m/s for two layouts S_1 and S_2 and about 0.10 m/s for human beings H_1, H_2 in the layout S_3 , respectively. The reason for the difference in maximum velocity is probably due to the fact that stabilised thermal plume (normally quite consist between two thermal human bodies) in the upper part of the flow domain in the layout S_3 , would cause the decrease of velocity magnitude. It is also noted that in the lower vertical regions of two sides of the small sofa, air velocity contours are quite similar between two layouts S_1 and S_3 with human being H_1 (see Figures 6(a), 6(c)). Around human being H_1 and the sofa, the velocity contours are relatively symmetrical, while around human being H_2 , the flow pattern is more complex. This is mainly due to the location of the occupant, e.g. the occupant who is close to the window wall is likely to be more affected by the inflow from the inlet and the thermal plume from the heat source (i.e. radiator and TV).



Figure 5.7. Air temperature distributions at a mid-streamwise plane (z = 0 m) for three layouts with sofa: (a) S_1 , (b) S_2 , (c) S_3 .



Figure 5.8. Velocity contours at the mid-width of occupant body for three layouts with sofa: at x = 4.25 m (a) S_1H_1 and (c) S_3H_1 and at z = 0.85 m (b) S_2H_2 and (d) S_3H_2 .

Figure 5.9 shows the corresponding PPD map around an occupant at four vertical locations Y = 0.1m, 0.5m, 1.1m and 1.3m for each sofa layout, based on the CFD predicted velocity, air temperature and radiation temperature at the eight measuring points on 1m-circle occupied zone (see Figure 5.3(b)). The PPD predictions are

presented in red lines for occupant H_1 and in blue lines for occupant H_2 , and in solid line for two layouts S_1 and S_2 and in dashed line for third layout S_3 , respectively. It is clear that the PPD distributions around an occupant in two sofa layouts S_1 and S_2 are resemblance throughout the domain heights, apart from a low position of Y = 0.5 m. The PPD magnitudes are also generally lower (below 10) for the layouts S_1 and S_2 , compared to that of layout S_3 . Note that small value of PPD < 10 is highly recommended as desirable environment for occupied spaces in terms of thermal comfort requirements (International Organization for Standardization, 2007). There are two peaks at positions 135° and 225° at Y = 0.5 m level (i.e. beside the hip) and one peak at position 180° at Y = 1.1 m (i.e. back of the neck) observed and their existences could be due to the location close to the surfaces of occupant and other adiabatic surfaces (e.g. the sofa and the walls), thus largely affected by elevated air temperature and radiation temperature and as well as the low velocity magnitude (almost zero), respectively. In the sofa layout S_3 , both occupants are having uncomfortable conditions throughout the vertical level in the domain, due to significant air temperature increases by 1.5 °C across the domain, compared with that of the volumetric fluid temperature in two layouts S_1 and S_2 . The similar findings were previously reported by Lin et al. (Lin et al. 2005). It can be also seen in the layout S_3 that there are large fluctuations in the PPD values for occupant H_2 at two lower vertical levels (Y = 0.1 m, 0.5 m) whilst at two higher vertical levels (Y =1.1 m, 1.3 m), its PPD predictions are aligned with that of the occupant H_1 . This is probably due to the fact that there is no big difference between two occupants in the upper part of the domain in terms of air temperature and flow velocity. Comparing with that of Myhren's study of unfurnished and unoccupied room (Myhren and Holmberg, 2009), it was found that the predicted PPD values in present study increase with the vertical height of the domain, while Myhren's results showed the decrease trend with the height. This discrepancy may be due to the existence of occupants in the domain, creating different flow paths and thermal plumes around them especially along the vertical direction.

As the occupant H_2 is more close to the window along the streamwise direction, it is more likely affected by the mixing effects of the cold inflow from the inlet and the warm thermal plume from the heat source (see Figure 5.7). This will cause non-uniform distributions of air temperature around the occupant H_2 especially in the region of lower vertical levels, resulting in asymmetric PPD distributions in the front face (i.e. at orientation of 0°). In contrary, the symmetrical PPD distributions are observed for occupant H_1 facing against the cold inflow and located at a further opposite wall to the window wall. Furthermore, at the two locations of occupants H_1 and H_2 , the temperature difference is quite large (about 1.6 °C) in the region of lower vertical levels, and becomes very small (about 0.2 °C) in the region of high vertical levels. There is little influence from the velocity field on the PPD calculation, since most of the velocity magnitude at the measuring points are very small, generally below 0.05 m/s.

Based on above analysis of indoor thermal comfort in a relaxing mode in the domain, it can be concluded that the PPD magnitude increases with a number of occupants in the room. For example the PPD magnitude would increase by 8.6 % in the layout S_3 , compared with single occupant of the layout S_1 . This would lead to uncomfortable condition for the occupants. In case of single occupant (e.g. the layout S_1 or the layout S_2), there is no major differences in terms of the level of thermal discomfort value (i.e. PPD value). Furthermore, it is found that thermal comfort indices are very sensitive to the orientation of the incoming flow stream path towards the occupant, particularly at the lower vertical levels. A near symmetrical PPD distribution is obtained in the spanwise direction against the main stream, while asymmetrical PPD distribution is observed in the streamwise direction against the main stream as seen in Figure 5.8 and Figure 5.9.



Figure 5.9. PPD predictions for seated occupants H_1 (in red lines) and H_2 (in blue lines) in different layouts with sofa (S_1 and S_2 in solid lines, while S_3 in dashed line) at four vertical locations (a) Y = 0.1 m, (b) Y = 0.5 m, (c) Y = 1.1 m and (d) Y = 1.3 m, respectively.

5.2.3. Effect of heat transfer from heat-generating furniture (Case 5.1.3)

Figure 5.10 shows the influence of heat-generating furniture such as a TV with "onmode" on the volume-average air temperature with the average PPD around two occupants (H_1 and H_2) for three layouts with sofa (S_1 , S_2 and S_3), comparing with that of the same layout but with TV "off-mode". Note that an average PPD value is calculated from eight measuring points surrounding an occupant throughout the four vertical levels Y = 0.1 m, 0.5 m, 1.1 m and 1.3 m. It can be seen that for a TV of 2000 W/m^3 heat generation, air temperature in the domain increases by approximately 4.4 – 5.2 %, due to the TV surface temperature of 31 - 33 °C, compared with that of TV "off-mode" in case 5.1.2. These changes cause the PPD value increases by 4.7 - 5.4 % for two layouts S_1 and S_2 and 11.1 - 11.5 % for third layout S_3 . As a result, thermal comfort level is mostly likely decreased, especially when the volume-average air temperature is above 24 °C.



Figure 5.10. Comparison of averaged PPD values around occupants (H_1, H_2) and volume-average air temperature for three layouts with sofa (S_1, S_2, S_3) with TV "on-mode" and "off-mode".

From the polar map of PPD values from these case studies (not shown here), the distribution shapes are similar to that of Figure 5.8, but the magnitudes are slightly higher than that of the acceptable indoor thermal condition. The predicted rate of PPD variation with temperature are in good agreement with published results of Lin *et al.* (Lin *et al*, 2005), e.g. 1 - 5% for 1 °C temperature increase and 21 - 27% for 3 °C temperature increase, respectively. An addition of heat generating furniture does not affect significantly on the PPD distributions, compared with "off-mode" of the TV.

5.3. Numerical optimisation of thermal comfort

The aim of indoor environment numerical optimisation study is to achieve better thermal comfort for occupants without changing the room layout. Based on studied above, further investigation continues by varying the heat generation magnitude of the heat

source and the ventilation flow rate, to understand their influences on the indoor temperature and environment. The parameters considered are $4000 < q_{rad}(W/m^3) <$ 4500 for the heat generation of heat source (i.e. radiator panels) and $0.5 < U_{inlet}(m/s) < 1.0$ for ventilation velocity. The heat generation of heat source is kept at medium heat level between 38 - 42 °C. The ventilation velocity is also within the requirements of indoor thermal comfort in dwellings (e.g. ventilation rate of 0.5 -1.0 ACH (Office of the Deputy Prime minister, 2006)). Note that ACH is air change per hour (air change rate) (1/h), representing the circulation frequency that the air within an enclosed space is replaced. This is equivalent to an air supply rate of 4.2 - 8.3 L/s in the model room of present study. Each case study applies to all three layouts with sofa S_1, S_2, S_3 . Based on the results obtained from case studies above, further two cases are conducted, i.e. heat generation study (case 5.3.2) and ventilation velocity study (case 5.3.3), see Table 5.4 for description. The results will be compared with case study above (i.e. case 5.3.1: $q_{rad} = 4500 W/m^3$, $U_{inlet} = 0.7 m/s$ and $q_{TV} = 2000 W/m^3$ with $T_{human} = 31^{\circ}$ C), in terms of the PPD and PMV values around the occupants, and fluid (air) temperature, respectively.

Table 5.4. I	Parametric	case	studies
--------------	------------	------	---------

Case	$q_{rad} \left(W/m^3 \right)$	U _{inlet} (m/s)	$q_{TV} (W/m^3)$
5.3.1 (Baseline study)	4500	0.7	2000
5.3.2 (Heat generation study)	4000	0.7	2000
5.3.3 (Ventilation velocity study)	4500	1.0	2000

Figure 5.11 shows the calculated PPD value around an occupant (H_1 and/or H_2) for case 5.3.2 and case 5.3.3 in three layouts with sofa at four vertical locations, compared with that of case 5.3.1. The PPD predictions are presented in red lines for the occupant H_1 and in blue lines for the occupant H_2 , and in dash-dotted-dotted line for case 5.3.1, solid line for case 5.3.2 and dashed line for case 5.3.3, respectively. Results from two layouts S_1 and S_2 are shown on the left-hand-side column, while that of the layout S_3 on the right-hand-side column in Figure 5.11. It is clear that both case 5.3.2 and case 5.3.3 successfully reduce the PPD level, compared with that of the previous baseline study (i.e. case 5.3.1). Results from case 5.3.3 has slightly better thermal environment than that of

case 5.3.2, but do not have significantly improvement. In case of single occupancy, i.e. (layout S_1 and layout S_2), the predicted PPD values are generally within 10 % variations throughout the vertical points, while in the layout S_3 , only case 5.3.3 gives desirable values (i.e. less than 10 %). Overall, the PPD value improves about 4.3 – 6.5 % for case 5.3.2 and 6.2 – 17.6 % for case 5.3.3 in the layout S_3 , compared with that of case 5.3.1. The large reduction of the PPD index is probably due to lower air temperature in the fluid domain, as shown in Figure 5.12.

Although the PPD predictions have shown some positive improvements of thermal comfort, thermal sensation (e.g. the way of feeling thermal comfort) could be divergent at the location of occupant. Figure 5.12 shows comparison of average PMV value and volume-average temperature for case 5.3.2 and case 5.3.3 for each occupant in the sofa layouts, compared with that of the baseline case 5.3.1. It is clear that case 5.3.1 predicts the highest PMV values for all three cases while case 5.3.3 gives the lowest predictions. In both case 5.3.1 and case 5.3.2, the occupants in the layout S_3 could feel uncomfortable with excess level of warm environment, (i.e. 0.5 < PMV). In contrary, the occupant in the layouts S_1 and S_2 in case 5.3.3 could feel slightly cooler or neutral, because of the predicted PMV value of -0.12 and 0.04, respectively. It can also be seen that there are noticeable volume-average temperature differences between case 5.3.1 and case 5.3.2 (around 0.8 °C) and between case 5.3.1 and case 5.3.3 (around 2.6 °C), while the average temperature of case 5.3.3 is below 24 °C for all three layouts. Two cases (i.e. case 5.3.1 and case 5.3.2 in the layout S_3) predicted the PPD value above 10 %, corresponding to the volume-average air temperature of greater than 24 °C. The impact of ventilation velocity increase on thermal comfort seems more significant, as the thermal transfer could be predominantly driven by higher ventilation rate from the inlet opening and thus affects the indoor environment. This has been confirmed from all case studies with the occupants and the sofa layouts.



Figure 5.11. PPD predictions for seated occupants H_1 (in red line) and H_2 (in blue line) in three layouts with sofa (S_1 and S_2 (a, c, e, g) and S_3 (b, d, f, h)) for case 5.3.1 (in dash-dotted-dotted lines), case 5.3.2 (in solid lines) and case 5.3.3 (in dashed lines) at four vertical locations (a, b) Y = 0.1 m, (c, d) Y = 0.5 m, (e, f) Y = 1.1 m and (g, h) Y = 1.3 m, respectively.



Figure 5.12. Comparison of average PMV values around occupants (H_1, H_2) and volumeaverage air temperature from three layouts with sofa (S_1, S_2, S_3) for cases 5.3.1, 5.3.2 and 5.3.3.

5.4. Summary

An investigation of indoor heat transfer in a 3-D furnished and occupied model room with localised heat source and window glazing has been carried out by computational fluid dynamics approach. The computational model was carefully validated against published data in literature for a 3-D empty model room (Myhren and Holmberg, 2008), (Myhren and Holmberg, 2009), (Olesen et al, 1980). The model has been used to investigate the effect of furniture arrangement with and without heat generation and occupants on indoor thermal comfort. After the investigation of the effect of non-heat generating furniture arrangement/location, heat generation furniture and human beings are introduced to the computational domain. It can be concluded in the analysis of the effect of thermal human being in a relaxing mode and furniture on indoor heat transfer in the domain that the presence of furniture (case 5.1.1) influences the flow pattern, resulting in the development of flow re-circulations around furniture. While the velocity magnitude is generally consistent at the monitoring location on a streamwise mid-plane throughout the cases, it is slightly higher in the vicinity of the furniture (e.g. see Figure

5.5). It seems that the buoyancy strength is perhaps not significantly affected by the presence of furniture, indicating that room temperature would be sustained at the similar level as that of an empty room layout S_0 . The presence of thermal occupant (case 5.1.2) influences indoor environment temperature field with the formation and development of thermal plume from the body, increasing volume-averaged temperature by maximum 15 %, compared with that of unoccupied and empty model room case S_0 . While having two occupants in the room (e.g. the layout S_3), air temperature in the entire domain would increase by 6.5 %, compared with the lowest volume-averaged temperature from single occupant of the layout S_1 . The impact of a TV "on-mode" in an occupied and heated model room (case 5.1.3) on thermal comfort around the occupants is not very significant, except having an increase of the PPD value around the occupants by maximum 5.4 % for one occupant and 11.5 % for two occupants.

An investigation of indoor thermal environment in a 3-D furnished and occupied model room with localised heat source and window glazing has been carried out by computational fluid dynamics approach. The computational model carefully validated against published data (Myhren and Holmberg, 2008), (Myhren and Holmberg, 2009), (Olesen et al. 1980) has been used to investigate the effect of furniture arrangement with and without heat generation and occupants on indoor thermal comfort. After the investigation of heat transfer in the thermally complex domain, thermal comfort level in case 5.1.2 and case 5.1.3 was assessed and improved towards the ISO recommended indoor environment condition. It is found that the PPD magnitude increases with a number of occupants, by 8.6 % in the layout S_3 in comparison with single occupant of the layout S_1 , towards the uncomfortable condition defined in the ISO standard. An addition of heat generating furniture does not affect significantly on the PPD distributions, compared with "off-mode" of the TV. The location of occupant is found very sensitive to flow stream path, e.g. the PPD distribution is symmetrical in the spanwise position but becomes asymmetrical in streamwise position. In a model room configuration as studied hereby, desirable indoor environment can be achieved under flow and thermal conditions of $q_{rad} = 4000 W/m^3$ and $U_{inlet} = 0.7 m/s$ and $q_{rad} = 4500 W/m^3$ and $U_{inlet} =$ 1.0 m/s with a fixed $q_{TV} = 2000 W/m^3$ for single occupancy sitting on a sofa watching TV. With more occupants introduced to the room, it is highly recommended that higher

ventilation flow rate $(U_{inlet} > 0.7 m/s)$ would be required to achieve desirable thermal conditions, rather than reducing heat generation from the heating sources.

Chapter 6

Conclusion and Future Work

6.1. Summary

The current trend of climate change, technology improvement and new building scheme results in a growing interest in the indoor environment. The study of indoor space is mainly the characteristics of flow and heat transfer and the health and comfort level. The problem is raised that the information of indoor thermal transfer in a space that occupant being relax (e.g. a domestic living room) is limited since indoor environment is very complex in geometry and flow nature and there is a large gap in the information, for example, between 2-D simple geometry and 3-D complex geometry.

Within this context, the aim of the present thesis was to provide the detailed information of indoor environment in a complex and realistic living space and investigate thermal comfort improvement. For this purpose, computational models were developed to analyse indoor thermal transfer affected by thickness of boundary, material property, location and size of heat source, window glazing and furniture, thermal loads, ventilation rate and flow oscillation. The numerical results were validated against available experimental and numerical data. The findings of the thesis are summarised below.

6.2. Conclusion

- (1) In the study of conjugate natural convection heat transfer in a ventilated room with localised heat source and window glazing,
 - (i) The sizes of heat source and glazing had significant impact on temperature field.
 - (ii) The heat energy loss through the solid wall surfaces (i.e. adjacent to the radiator panels) was about 35 % when reducing the wall thickness by 10 cm from the original wall thickness 30 cm to 20 cm. This reduced the volume-averaged

thermal comfort temperature by 9 % compared with that when original wall thickness.

- (iii) The large amount of heat loss was mainly influenced by the heat source being next to the solid wall without suitable insulation.
- (iv) With the minimum wall thickness to meet UK's domestic house requirements, the thermal comfort could be sustained within the indoor environment standards. However, the total heat loss through a thinner wall of 20 cm thickness was about 53 % high, compared with that through a thicker wall of 40 cm thickness
- (v) Ideal indoor thermal environment could be achievable with a radiator size of $h_B/H = 0.23 0.31$, window glazing size of $h_2/H = 0.38 0.46$, wall thickness of d/L = 0.042 0.063, and thermal conductivity ratios of $k_{air}/k_{wall} = 0.08 0.28$, respectively. The configuration of thinner wall d/L = 0.042 and wall thermal conductivity of $k_{air}/k_{wall} = 0.28$ could be applied to the region that has warmer winter conditions. However for cold winter conditions, a large size radiator panel, well-insulated walls, and a low wall thermal conductivity would be required.
- (2) In the study of indoor environment in a 3-D furnished and occupied model room with localised heat source and window glazing,
 - (i) The presence of furniture influenced the flow pattern, resulting in the development of flow re-circulations around furniture.
 - (ii) The velocity magnitude was slightly higher in the vicinity of the furniture but it was generally consistent at the monitoring location on a streamwise mid-plane throughout the cases.
 - (iii) The buoyancy strength was not significantly affected by the presence of furniture, indicating that room temperature would be sustained at the similar level as that of an empty room.
 - (iv) The presence of thermal occupant influenced indoor environment temperature field with the formation and development of thermal plume from the body, increasing volume-averaged temperature by maximum 15%, compared with that of unoccupied and empty model room.

- (v) With two occupants in the room, air temperature in the entire domain would increase by 6.5 %, compared with the lowest volume-averaged temperature from single occupant.
- (vi) The impact of a TV "on-mode" in an occupied and heated model room on comfort temperature around the occupants was not very significant, except having an increase of the averaged PPD value around the occupants by maximum 5.4 % for one occupant and 11.5 % for two occupants.
- (3) In the corresponding thermal comfort level using Fanger's indices in a 3-D furnished and occupied model room with localised heat source and window glazing,
 - (i) The PPD magnitude increased with a number of occupants, by 8.6 % in the double occupants in comparison with single occupant, towards the uncomfortable condition defined in the ISO standard.
 - (ii) An "on-mode" TV did not affect significantly on the PPD distributions, compared with "off-mode" of the TV.
 - (iii) The location of occupant was very sensitive to flow stream path, e.g. the PPD distribution was symmetrical in the spanwise position but became asymmetrical in streamwise position.
 - (iv) Desirable indoor environment might be achieved under flow and thermal conditions of $q_{rad} = 4000 W/m^3$ and $U_{inlet} = 0.7 m/s$ and $q_{rad} = 4500 W/m^3$ and $U_{inlet} = 1.0 m/s$ with a fixed $q_{TV} = 2000 W/m^3$ for single occupancy sitting on a sofa watching TV.
 - (v) With more occupants introduced to the room, the higher ventilation flow rate $(U_{inlet} > 0.7 \text{ m/s})$ would be required to achieve desirable thermal conditions, rather than reducing heat generation from the heating sources.
- (4) In the flow oscillation study in 3-D ventilated model room,
 - (i) In the forced convection flow in an empty-square domain, no direct relation between velocity and turbulent flow in power spectral density and frequency was found. Each of time-history velocity oscillations was independent and random.

- (ii) The dominant frequencies increased its value as an unheated box was placed at the centre of the domain but decrease with heat on it.
- (iii) The existence of a box divided frequency into two different values, concluding that dependence of frequency magnitude and fluctuation patterns on location against stream direction: a high energy at a lower frequency on a spanwise plane while a low energy at a higher frequency on a streamwise plane.
- (iv) The flow oscillation above the box was in irregular manner, caused by the recirculations above the top surface of the box and between the rear box surface and domain back wall.
- (v) With the thermal box, the frequency of velocity oscillation was consistent with temperature at the location although the energy of the fluctuation is much higher in temperature, especially on a spanwise mid-plane.
- (vi) The formation of thermal plume from the heated box stabilised flow in the upper part (e.g. high Grashof number of 0.5×10^6 (Sinha et al, 2000)) and the sides of the heated box on a spanwise plane, causing to lower the dominant frequencies.

6.3. Contributions

This study of indoor thermal environment with regard to the thermal comfort evaluation has contributed in a number of ways by the flow and heat transfer effects due to conjugate natural convection, furniture arrangement and occupant number, and flow oscillations. Its main contributions are summed up as follows.

- Built relations between conductive heat transfer and room structure
- Secured occupant feeling in different locations and thermal scenarios
- Detailed correlation between heat transfer and flow unsteadiness

The current study summarises some of the important reservations with regard to the CFD capability and reliability for indoor thermal environment and present data would be useful for the built environment thermal engineers in design and optimisation of domestic rooms.

6.4. Future Work

Indoor thermal comfort has been increasingly important in built environment. The study of the topic is still to be further development in many aspects. Recommendations for further improvements and future research are given below.

- (1) Models always need to be as realistic complex as possible. An investigation of empty room does not account for the detailed airflow and heat transfer inside the domain. For expanding the model studied in the thesis, it can be recommended to consider of lights on the ceiling, an open door, etc.
- (2) In energy assessment, it might be useful to calculate using Integrated Environmental Solutions (IES) or TAS from Environmental Design Solutions Limited (EDSL) with for example the Building Regulations Part L. The software is widely adopted in indoor energy consultant and industry in the UK. This will help widen the student's knowledge of indoor environment for their future career.

References

Alahmer, A., Abdelhamid, M., Omar, M. Design for thermal sensation and comfort states in vehicles cabins. Applied Thermal Engineering 2012; 36: 126-140.

Alfano, F. R. D., Ianniello, E., Palella, B. I. PMV-PPD and acceptability in natural ventilated schools. Building and Environment 2013; 67: 129-137.

Alfano, F. R. D., Palella, B. I., Riccio, G. The role of measurement accuracy in the thermal environment assessment by means of OMV index. Building and Environment 2011; 46: 1361-1369.

Andreasi, W. A., Lamberts, R., Candido, C. Thermal acceptability assessment in buildings located in hot and humid regions in Brazil. Building and Environment 2010; 45: 1225-1232.

ANSYS 13.0 Fluent. User's guide: Radiation temperature. 2006.

Antonelli, M., Mazzino, A., Rizza, U. Statistics of temperature fluctuations in a buoyancy-dominated boundary layer flow simulated by a large eddy simulation model. Journal of Atmospheric Sciences 2003; 60: 215-224.

Atmaca, I., Kaynakli, O., Yigit, A. Effects of radiant temperature on thermal comfort. Building and Environment 2007; 42: 3210-3220.

Barbhuiya, S., Barbhuiya, S. Thermal comfort and energy consumption in a UK educational building. Building and Environment 2013; 68: 1-11.

Barrington, A. Domestic energy bills in 2012: The impact of variable consumption. Department of Energy & Climate Change. 2013. <u>www.gov.uk</u> [accessed April 2014]

Ben Nasr, K., Chouikh, R., Kerkeni, C., Guizani, A. Numerical study of the natural convection in cavity heated from the lower corner and cooled from the ceiling. Applied Thermal Engineering 2006; 26: 772-775.

Betts, P. L., Bokhari, I. H. Experiments on turulent natural convection in an enclosed tall cavity. Heat and Fluid dlow 2000; 21: 675-683.

Beya, B. B., Lili, T. Oscillatory double-diffusive mixed convection in a two-dimensional ventilated enclosure. International Journal of Heat and Mass Transfer 2007; 50: 4540-4553.

Bilgen, E. Conjugate heat transfer by conduction and natural convection on a heated vertical wall. Applied Thermal Engineering 2009; 29: 334-339.

Blay, D., Mergui, S., Niculae, C. Confined turbulent mixed convection in the presence of horizontal bupyancy wall jet. Fundamentals of mixed convection 1992; 213: 65-72.

Boermans, T., Petersdorff, C. Summary of U-values for better energy performance of buildings. ECOFYS. 2007.

Bojic, M., Yik, F., Lo, T. Y. Locating air-conditioners and furniture inside residential flats to obtain good thermal comfort. Energy and Buildings 2002; 34: 745-751.

Bos, M. A., Love, J. A. A field study of thermal comfort with underfloor air distribution. Building and Environment 2013; 69: 233-240.

Buratti, C., Moretti, E., Belloni, E., Cotana, F. Unsteady simulation of energy performance and thermal comfort in non-residential buildings. Building and Environment 2013; 59: 482-491.

Cappelletti, F., Prada, A., Romagnoni, P., Gasparella, A. Passive performance pf glazed components in heating and coolinf of an open-space office under controlled indooe thermal comfort. Building and Environment 2014; 72: 131-144.

Cengel, Y. A., Ghajar, A. J. Heat and mass trasnfer: Fundamentals and applications furth edition in SI units. New York: McGraw-Hill. 2011.

Chandrasekhar, S. Radiative Transfer. New York: Dover Publications. 1960.

Chen, Q. Comparison of different k-e models for indoor air flow computions. Numerical Heat Transfer, Part B 1995; 28: 353-369.

Chen, Q. Prediction of room air motion by Reynolds-stress models. Building and environment 1996; 31: 233-244.

Chen, Q., Lee, K., Mazumdar, K., Poussou, S., Wang, L., Wang, M., Zhang, Z. Ventilation performance prediction for buildings: Model assessment, Building and Environment 2010: 45: 295-303.

Chen,Q., Srebric, J. How to verify, validate, and report indoors environment modeling CFD analyses. ASHRAE TC 4.10, Indoor Environmental Modeling. ASHRAE. 2001.

Chu, H. H. S., Churchill, S. W., Patterson, C. V. S. The effect of heater size, location, aspect ratio, and boundary conditions on two-dimensional, laminar, natural convection in rectangular channels. Journal of heat transfer 1976; 98: 194-201.

Cooley, J. W., Tukey, J. W. An algorithm for the machine calculation of complex Fourie series. Mathematics of Computation 1965; 19: 297-301.

Dascalaki, E., Santamouris, M., Argiriou, A., Helmis, C., Asimakopoulos, D., Papadopuolos, K., Soilemes, A. Predicting simgle sided natural ventilation rates in buildings. Solar Energy 1995; 55: 327-341.

Deng, Q. H., Tang, G. F. Numerical visualization of mass and heat trasnfer for conjugate natural convection/heat conduction by streamline and heatline. International Journal of Heat and Mass transfer 2002; 45: 2373-2385.

Department for Communities and Local Government. Building regulations: Energy efficiency requirements for new dwellings. London. 2007. <u>www.gov.uk</u> [accessed October 2012]

Department for Communities and Local Government. English Houseing Survey: Housing stock report 2008. London. 2010. www.gov.uk [accessed April 2011]#

Department of Energy and Climate Change. Energy Consumption in the UK (2014): Chapter 1 Overall energy consumption in the UK since 1970. <u>www.gov.uk</u> [accessed February 2015]

Department of Trade and Industry. Energy consumption in the United Kingdom. London:National Statistics. 2002. <u>www.gov.uk</u> [accessed September 2012]

Deuble, M. P., de Dear, R. J. Mixed-mode buildings: A double standard in occupants' comfort expectations. Building and Environment 2012; 54: 53-60.

Energy saving trust. Roof and loft insulation 2013. <u>www.energysavingtrust.org.uk</u> [accessed June 2013] European Commission, Directorate-General for Energy and Transport. Saving 20% by 2020 - Action plan for energy efficiency:realising the potential. 2006. <u>ec.europa.eu</u> [accessed June 2013]

European Comission. Low energy building in Europe: Current state of play, definitions and best practice. Brussels. 2009. <u>ec.europa.eu</u> [accessed August 2010]

European Commision. EU energy policy for buildings after the recast, Unit D4 – Energy efficiency, DG TREN. Directorate General for Energy and Transport. <u>ec.europa.eu</u> [accessed May 2014].

Fanger, P. O. Thermal comfort. New York: McGraw Hill. 1972

Fanger, P. O., Pedersen, C. J. K. Discomfort due to air velocities in spaces. Proceeding of the meeting of Commissions B1, B2, E1 of the IIR, 4, pp. 289-296. Belgrade, Yugoslavia. 1977.

Fanger, P. O., Toftum, J. Extension of the PMV model to non-air-conditioned buildings in warm climates. Energy and Buildings 2002; 34: 533-536.

Federspiel, C. C., Asada, H. User-adaptable comfort control for HVAC systems. Journal of Dynamic Systems, Measurement, and Control 1994; 116: 474-486.

Fluent. Introduction to the Renormalization group method and turbulence modeling. Technical Memorandum TM-107. Lebanon, USA: Fluent Inc. 1993.

Frontczak, M., Wargocki, P. Literature survey on how different factors influence human comfort in indoor environments. Building and Environment 2011; 46: 922-937.

Gagge, A. P., Stolwijk, J. A. J., Nishi, Y. An effective temperature scale based on a simple model of human physiological regulatory response. ASHRAE Transactions 1971; 77: 247-262.

Gan, G. General expressions for the calculation of air flow and heat transfer rates in tall ventilation cavities. Building and Environment 2011; 46: 2069-2080.

Gebhart, B., Mahajan, R. Chatacteristic distrubance frequency in vertical natural convection flow. International Journal of Heat and Mass Transfer 1975; 18: 1143-1148.

Hanzawa, H., Melikow, A. K., Fanger, P. O. Field measurements of characteristics of turbulent air flow in the occupied zone of ventilated spaces. Fanger, P. O. ed: CLIMA

2000: Proceeding of the world congress on heating, ventilating and air-conditioning, 4, pp. 409-414. Copenhagen, Denmark. 1985.

Henkes, R. A. W. M., Hoogendoorn, C. J. Bifurcation to unsteady natual convection for air and water in a cavity heated from the side. Proceeding 9th International heat ans transfer conference, 2, pp. 257-262. Jerusalem, Israel. 1990.

Hens, H. S. Thermal comfort in office buildings: Two case studies commented. Building and Environment 2009; 44: 1399-1408.

International Organization for Standardization, ISO 13370. Thermal performance of buildings - Heat transfer via the ground - Calculation methods. 2007. <u>www.iso.org</u> [accessed November 2012]

Homod, R. Z., Sahari, K. S. M., Almurib, H. A. F., Nagi, F. H. RLF and TS fuzzy model indentification of indoor thermal comfort based on PMV/PPD. Building and Environment 2012; 49: 141-153.

Horikiri, K. Yao, Y., Yao, J. Numerical simulation of convective airflow in an empty room. International Journal of Energy and Environment 2011; 5: 574-581.

Horikiri, K., Yao, Y., Yao, J. Numerical study of unstead airflow phenomena in a ventilated room. Computational Therml sciences 2012; 4: 317-333.

Horikiri, K., Yao, Y., Yao, J. Modelling conjugate flow and heat transfer transfer in a ventilated room for indoor thermal comfort assessment. Building and Environment 2014; 77: 135-147.

Jang, M. S., Koh, C. D., Moon, I. S. Review of thermal comfort design based on PMV/PPD in cabins of Korean amritime patrol vessels. Building and Environment 2007; 42: 55-61.

Jensen, O. M., Wittchen, K. B., Thomsen, K. E. Towards very low energy buildings. EuroACE, The European Aliance of Companies for Energy Efficiency of Buildings. Danish Building Research Institute. 2009.

Jiang, Y., Alexander, D., Jenkins, H., Arthur, R., Chen, Q. Natural ventilation on in buildings: Measurement in a wind tunnel and numerical simulation with large eddy simulation. Journal of Wind Engineering and Industrial Aerodynamics 2003; 91: 331-353. Jiang, Y., Chen, Q. Buyoncy-driven single-sided natural ventiltion. International journal of heat and mass transfer 2003; 46: 973-988.

Jin, M., Zuo, W., Chen, Q. Simulating natural ventilation in and around buildings by fast fluid dynamics. Numarical Heat transfer, Part A: Applications, An International Journal of Computation and Methodology 2013; 64: 273-289.

Kaluri, R. S., Basak, T., Roy, S. Heatline approach for visualization of heat flow and efficient thermal mixing with discrete heat sources. International Journal of Heat and Mass transfer 2010; 53: 3241-3261.

Kaminski, D. A., Prakash, C. Conjugate natural convetion in a square enclosure: effect of conductioin in one of the vertical walls. International journal of heat and mass transfer 1986; 29: 1979-1988.

King, K. J. Turbulent natural convection in rectangulr air cavities. PhD thesis, Queen Mary and Westfield College, University of London, London, UK. 1989.

Koca, A. Numerical analysis of conjugate heat trasnfer in a partially open square cavity with a vertical heat source. International Communications in Heat and Mass Transfer 2008; 35: 1385-1395.

Kovanen, K., Seppane, O., Siren, K., Majanen, A. Turbulent air flow measurements in ventilated spaces. Environment International 1989; 15: 621-626.

Krajcik, M., Simone, A., Olesen, B. W. Air distribution and ventilation effectiveness in an occupied room heated by warm air. Energy and Buildings 2012; 55: 94-101.

Kumar, A., Suman, B. M. Experimental evaluation of insulation materials for walls and roofs and their impact on indoor thermal comfort under composite climate. Builsing and Environment 2013; 59: 635-643.

Kuznetsov, G. V., Sheremet, M. A. Conjugate natural convection in an enclosure with a heat source of constant heat transfer rate. International Journal of Heat and Mass Transfer 2011; 54: 260-268.

Lan, L., Pan, L., Lian, Z., Huang, H., Lin, Y. Experimental study on thermal comfort of sleeping people at different air temperatures. Building and Environment 2014; 73: 24-34.

Launder, B. E., Spalding, D. B. Lectures in mathematical models of turbulence. London: Academic Press. 1972. Li, F., Liu, J., Pei, J., Lin, C. H., Chen, Q. Experimental study of gaseous and particulate contaminants distribution in an aircraft cabin. Atomospheric Envrionment 2014; 85: 223-233.

Lin, Z., Chow, T. T., Tsang, C. F., Chan, L. S., Fong, K. F. Effect of air supply temperature on the performance of displacement ventilation (Part 1) - Thermal comfort. Indoor and Building Environment 2005; 14: 103-115.

Lu, W., Howarth, A. T., Jeary, A. P. Prediction of airflow and temperature field in a room with convective heat source 1997; 32: 541-550.

MacArther, J. W. Humidity and predicted-mean vote-base (PMV-base) comfort control. ASHRAE Transactions 1986; 92: 5-17.

Menter, F. R. Two-equation eddy-viscosity turbulence models for engineering applications. AIAA Journal 1994; 32: 1598-1605.

Muftuoglu, A., Bilgen, E. Conjugate heat transfer in open cavities with a discrete heater at its optimized position. International Journal of Heat and Mass Transfer 2008; 51: 779-788.

Myhren, J. A., Holmberg, S. Flow patterns and thermal comfort in a room with panel, floor and wall heating. Energy and Buildings 2008; 40: 524-536.

Myhren, J. A., Holmberg, S. Design considerations with ventilation-radiators: Comparison to traditiona; two-panel radiators. Energy and Buildings 2009; 41: 92-100.

Nationwide. House prices 2008. www.nationwide.co.uk [accessed June 2014].

Nilsson, H. O. Comfort climate evaluation with thermal manikin methods and computer simulation models. PhD thesis, KTH Royal Institute of Technology, Sweden. 2004

Niu, J. L., Zuo, H. G., Burnett, J. Simulation methodology of radiant cooling with elevated air movement. Building simulation, Seventh International IBPSA Conference. Rio de Janeiro, Brazil. 2001.

Ntibarufata, E., Hasnaoui, M., Vasseur, P., Bilgen, E. Numerical study of natural convection in habitat with direct gain passive window systems. Renewable energy 1994; 4: 33-40.

Nouanegue, H., Muftuoglu, A., Bilgen, E. Conjugate heat transfer by natural convection, conduction and radiation in open cavities. International Journal of Heat and Mass Transfer 2008; 51: 6054-6062.

Office of the Deputy Prime minister. The Building Regulations 2000, Ventilation. London: RIBA Bookshops. 2006.

Olesen, B. W., Mortensen, E., Thorshauge, J., Berg-Munch, B. Thermal comfort in a room heated by different methods. Los Angeles: ASHRAE Transactions 86. 1980.

Ozgokmen, T. M., Iliescu, T., Fischer, P. F. Reynodles number dependence of mixing in a lock-exchange system from direct numerical and large eddy simulation. Ocean Modelling 2009; 30: 190-206.

Palmer, J., Cooper, I. United Kingdom, housing energy fact file. Department of energy and climate change. 2012. <u>www.gov.uk</u> [accessed June 2013]

Park, A. H., Chung, W. J., Yeo, M. S., Kim, K. W. Evaluation of the thermal performance of a Thermally Activated Building System (TABS) according to the thermal load in a residential building. Energy and Buildings 2014; 73: 69-82.

Patankar, S. V., Spalding, D. B. A calculation procedure for heat, mass and momentum transfer in three-dimensional parabolic flow. International Journal of Heat and Mass Transfer 1972; 15: 1787-1806.

Peach, J. Radiators and convectors. Journal of institution of heat and ventilating engineers 1972: 39: 239-253.

Pereira, M. L., Graudenz, G., Tribess, A., Morawska, L. Determination of particle concentration in the breathing zone for four different types of office ventilation systems. Building and Environment 2009; 44: 904-911.

Pourshaghaghy, A., Omidvari, M. Examination of thermal comfort in a hospital using PMV-PPD model. Applied Ergonomics 2012; 43: 1089-1095.

Raji A, Hasnaoui M, Bahlaoui A. Numerical study of natural convection deminated heat transfer in a ventilated cavity: Case of forced flow playing simultaneous assisting and opposing roles. International Journal of Heat and Fluid Flow 2008; 29: 1174-1181.

Restivo, A. Turbulent flow in ventilated rom. Ph.D. Thesis, University of London, Imperial College of Science and Technology, Mechanical Engineering Department, London. 1979.

Rim, D., Novoselac, A. Ventilation effectiveness as an indicator of occupant exposure to particles from indoor sources. Building and Environment 2010; 45: 1214-1224.

Saeid, N. H. Conjugate natural convection in a vertical porous layer sandwiched by finite thickness walls. International Communications in Heat and Mass Transfer 2007; 34: 210-216.

Scheatzle, D. G. The development of PMV-based control for residence in a hot and arid climate. ASHRAE Transactions 1991; 97: 1002-1019.

Selens, D., Parys, W., Baetens, R. Energy and comfort performance of thermally activated building systems including occupant behaviour. Building and Environment 2011; 46: 835-848.

Shih, Y. C., Chiua, C. C., Wang, O. Dynamic airflow simulation within an isolation room. Building and Environment 2007; 42: 3194-3209.

Shih, T. H., Liou, W. W., Shabbir, A., Yang, Z., Zhu, J. A new k-e eddy viscosity model for high Reynolds number turbulent flows. Computers and Fluids 1995; 24: 227-238.

Sinha, S. L., Arora, R. C., Roy, S. Numerical simulation of two-dimensional room air flow with and without buoyancy. Energy and Buildings 2000; 32: 121-129.

Smagorinski, J. General ciculation experiments with the primitive equations. Monthly weather review 1963; 91: 261-341.

Sorensen, D. N., Voigt, L. K. Modelling flow and heat transfer around a seated human body by computational fluid dynamics. Building and Environment 2003; 38 (6): 753-762.

Thorshauge, J. Air-velocity fluctuations in the occupied zone of ventilated spaces. ASHRAE Transactions 1982; 2: 753-764.

Tian, Y. S., Karayiannis, T. G. Low turbulence natural convection in an air filled square cavity Part 2: the turbulence quantities. Heat and mass transfer 2000; 43: 867-884.

Tye-Gingras, M., Gosselin, L. Comfort and energy comsumption of hydronic heating radiant ceilings and walls based on CFD analysis. Building and Environment 2012; 54: 1-13.

Wang M, Chen Q. Assessment of various turbulence models for transitional flows in enclosed environment (RP-1271). HVAC & R Research 2009; 15:1099-1119.

Wei, S., Li, M., Lin, W., Sun, Y. Parametric studies and evaluations of indoor thermal environment in wet season using a filed survey and PMV-PPD method. Energy and Buildings 2010; 42: 799-806.

Yakhot, V., Orszag, S. A. Renomalization group analysis of turbulence. I: Basic theory. Journal of Scientific Computing 1986; 1: 3-51.

Zhang, Z., Zhai, J. Z., Chen, Q. Evaluation of various CFD models in predicting room airflow and turbulence. 10th International conference on air distribution in rooms. Helsinki. 2007.

Zhang, Z., Zhang, W., Zhai, Z., Chen, Q. Evaluation of various turbulence models in predicting airflow and turbulence in enclosed environments by CFD: Part-2:Comparison with experimental data from literature. HVAC&R Research 2007; 13: 871-886.

Zhuang, R., Li, X., Tu, J. CFD study of the effects of furniture layout on indoor air quality under typical office ventilation schemes. Building Simulation 2014; 7: 263-275.

Zuo, W. Chen, Q. Real-time or faster-than-real-time simulation of airflow in buildings. Indoor air 2009; 19: 33-44.