A novel cabin air distribution strategy for automotive applications

Lim, Chun Wheng

2015

Lim, C. W. (2015). A novel cabin air distribution strategy for automotive applications.
Master's thesis, Nanyang Technological University, Singapore.

<https://hdl.handle.net/10356/62173>

<https://doi.org/10.32657/10356/62173>

**NANYANG
TECHNOLOGICAL
UNIVERSITY**

A Novel Cabin Air Distribution Strategy for Automotive Applications

Lim Chun Wheng

School of Mechanical and Aerospace Engineering

2015

A Novel Cabin Air Distribution Strategy for Automotive Applications

Lim Chun Wheng

School of Mechanical and Aerospace Engineering

A thesis submitted to the Nanyang Technological University
in partial fulfilment of the requirements for the degree of
Masters of Engineering

2015

Acknowledgments

I would like to thank my supervisor A/P Ooi Kim Tiow for his dedicated guidance and advice during this period for the project. Most importantly, I greatly appreciate the effort and time he spent sharing non-academic lessons, such as life experiences. The project period has been not only enriching in terms of the honing of technical and academic skills, I have learnt a lot more from him outside this context.

Secondly, I would like to thank my wife who has been very supportive during this period. Her patience, understanding and assistance gave me ample time to work on the project despite my busy work schedule. Her encouragement also helped me to push on and complete this project.

Abstract

This report looks at the current state of the art for air distribution methods in particular, personalized ventilation (PV) type of air distribution. PV systems have been widely used in the buildings applications and lately, in the development of new aircraft cabin air distributions. Air distribution in vehicles has had a few new cooling methods like spot cooling being researched on. PV systems have the aim of providing effective cooling with increased comfort providing better quality air to the occupant.

The solar heat load on a vehicle was studied. It was shown that the peak heat load in the vehicle does not occur when the sun is at its peak, but only when the sunlight is able to penetrate most of the windows at about 2 pm. There are also various methods to lower the direct solar radiation coming into the vehicle and one of them is via geometry. The best geometrical compromise to minimise solar radiation, and yet not affect the aerodynamics of the vehicle too drastically, is to position the windscreen at an angle of 40° from the vertical plane. This gives a reduction of about 15% in terms of solar loading compared to an angle of a conventional car at 57° . The most effective way of heat reduction is to prevent the solar radiation from entering the vehicle directly. If the peak cooling load and the energy usage for cooling were to be reduced from the cooling down phase after parking, some effective methods would be to allow for natural or forced ventilation of the ambient air during parking or lowering the thermal capacity of the materials in the cabin. Both methods can lead to a decrease in energy requirement by 27%.

A PV system for vehicles was introduced. This design was developed from prior designs which had various issues with airflow being affected by the warm air in the vehicle. The PV seat allowed for the individual cooling of a passenger. A series of simulations were conducted using StarCCM and Theseus FE. Three cases were ran, one being the full conventional air conditioning system with a mass flow rate of 0.1 kg/s , the next is a conventional air conditioning system but only with mass flow rate of 0.025 kg/s and focused entirely on the driver, and lastly the PV seat with the driver seated inside. The same mass flow rate was used for the PV seat case. All cases used an inlet air temperature of 8°C .

Comparing to latter 2 cases, the PV seat was able to produce a better comfort index score compared to the conventional seat with an improvement of about 0.7% to 50%. This was done with a reduction of air inlet velocity of 110%. The streamlines also showed that the airflow was more effective and much of the air did not flow into the back seat area reducing the cross contamination between passengers. Taking reference from breathing region at the back seat, measuring the spread of contaminants from the front, the PV seat case had 32% less contaminants compared to the convention air conditioning layout. Most importantly, although not proven via simulations, it can be deduced that potential energy savings are possible for a comparable comfort between the 2 types of cooling systems.

Contents

Acknowledgments.....	i
Abstract	ii
List of Abbreviations	vi
1 Introduction	1
1.1 Background	1
1.2 Objective	1
1.3 Scope.....	2
2 Literature review.....	4
2.1 Current air distribution methods	4
2.1.1 Automotive air distribution method.....	4
2.1.2 Conventional Building air distribution strategies.....	6
2.2 Personalised ventilation.....	8
2.2.1 Building personalized ventilation methods	10
2.2.2 Aircraft Personalised Ventilation Methods.....	15
2.2.3 Vehicle personalized ventilation methods.....	20
2.2.4 ATD Nozzle Design.....	24
2.3 Thermal Comfort Models.....	28
3 Simulation Governing Equations.....	32
3.1 Theory for heat transfer.....	32
3.2 Basic Governing Equations for CFD.....	38
3.2.1 Turbulence Models	41
3.3 Theory of on Virtual Human Manikin.....	44
3.3.1 Passive System	44
3.3.2 Active systems.....	52
3.3.3 Thermal comfort and sensations	54
3.3.3.5 Overall thermal sensation index	57
3.3.3.6 Local thermal comfort index	58
3.3.3.7 Overall thermal comfort index	59
4 Simulation of solar loading on a vehicle	60
4.1 Simulation setup	60
4.1.1 Changing of the heat load profile throughout the day.....	62
4.1.2 Heat loads at the different phases of vehicle operation	63
4.1.3 How various methods can help in the heat load control of the vehicle	64

4.1.4	How the change in the vehicle geometry can affect the heat load on the vehicle	65
4.1.5	Windows Properties.....	65
4.2	Results and Discussion	67
4.2.1	Changing of the heat load profile throughout the day	67
4.2.2	Heat loads at the different phases of vehicle operation	69
4.2.3	How various methods can help in the heat load control of the vehicle	72
4.2.4	How the change in the vehicle geometry can affect the heat load on the vehicle	74
4.2.5	Windows properties.....	75
5	Simulation of the PV Seat.....	77
5.1	CFD Co-Simulation.....	77
5.1.1	Tools	77
5.1.2	Cases to be simulated	78
5.2	Calibration Case	79
5.2.1	Case setup and boundary conditions.....	79
5.2.2	Results	81
5.2.3	Discussion.....	82
5.3	Design methodology of PV seat	83
5.3.1	Idea 1.....	83
5.3.2	Idea 2.....	86
5.3.3	Idea 3.....	88
5.3.4	Final design.....	91
5.4	Simulated cases for PV in vehicle.....	94
5.4.1	Case setup and boundary conditions.....	94
5.4.2	Physical models used in StarCCM simulations.....	98
5.4.3	Case 1: Conventional Air Conditioning at Medium Fan Speed	98
5.4.4	Case 2: Conventional Air Conditioning at Medium Fan Speed with air conditioning directed at only one person.....	99
5.4.5	Case 3: PV Seat Air Conditioning at Medium Fan Speed for only one person.	100
5.5	Results.....	101
5.5.1	Case 1: Conventional Air Conditioning at Medium Fan Speed	101
5.5.2	Case 2: PV seat with inlet velocity of 0.4 m/s with temperature of 8°C	111
5.5.3	Case 3: PV seat with inlet velocity of 0.84 m/s with temperature of 8°C ...	122

5.6	Discussion.....	133
5.6.1	Comfort	133
5.6.2	Comparison between Case 2 and 3.....	140
5.6.3	Energy usage and savings.....	146
6	Conclusions	148
7	Future Work	151
8	References.....	152
Appendix A	Vehicle Components	157
Appendix B	Vehicle materials.....	159
Appendix C	Calibration case with Experimental VS Simulation Plot.....	160
Appendix D	Plane slice for results plot for CFD modelling	173

List of Abbreviations

ANSYS	<i>(Software company which designed CFX & FLUENT)</i>	MP	<i>Moveable Panel</i>
ASHRAE	<i>American Society of Heating, Refrigeration, and Air-Conditioning Engineers</i>	MV	<i>Mixing Ventilation</i>
ATD	<i>Air Terminal Devices</i>	PEE	<i>Personal Exposure Effectiveness</i>
BMW	<i>(Brand of Car Manufacturer)</i>	PEM	<i>Personal Environment Module</i>
CAD	<i>Computer Aided Design</i>	PMV	<i>Predicted Mean Vote</i>
CFD	<i>Computational Fluid Dynamics</i>	PPD	<i>Predicted Percentage Dissatisfied</i>
CFX	<i>(Commercial CFD package by ANSYS)</i>	PV	<i>Personal ventilation</i>
CMP	<i>Computer Monitor Panel</i>	RadTherm	<i>(Commercial heat analysis software)</i>
COP	<i>Coefficient of Performance</i>	RC	<i>Re-circulated</i>
DMF	<i>Desk Mounted Fan</i>	RMP	<i>Round Moveable Panel</i>
EHT	<i>Equivalent Homogenous Temperature</i>	SBS	<i>Sick Building Syndrome</i>
FLUENT	<i>(Commercial CFD package by ANSYS)</i>	StarCCM+	<i>(Commercial CFD package)</i>
FORTTRAN	<i>(Programming language)</i>	Theseus FE	<i>(Commercial heat analysis software)</i>
HDG	<i>Horizontal Desk Grill</i>	TUM	<i>Technische Universität München</i>
HVAC	<i>Heating, Ventilation and Air Conditioning</i>	UCB	<i>University of California, Berkeley</i>
IR	<i>Infra-red</i>	UFV	<i>Under Floor Ventilation</i>
ISO	<i>International Organization for Standardization</i>	VDG	<i>Vertical Desk Grill</i>
		VOC	<i>Volatile Organic Compounds</i>

1 Introduction

1.1 Background

Electric vehicles have started to move away from being a research project to a commercial product. Despite huge advances in the development in the control and drive trains of these vehicles, energy consumption and battery capacity remains a serious issue. Advances in the energy storage such as batteries have progressed slowly over the years, but not quick enough to match energy requirements in an electrified world.

Much effort has been in developing more efficient devices and algorithms for control. Better vehicle design, use of regenerative braking and light yet stronger materials are areas that have been researched to create a next generation, energy efficient electric vehicle. However, one area that has been often considered a large consumption of energy, but yet often overlooked, is the air conditioning. It has been known that cooling the vehicle can take up to 20% of the energy used by the vehicle (Farrington & Rugh, 2000). If the air conditioning used lesser energy and the thermal load in the cabin lowered, this would allow the battery pack to have more capacity, and driving range would increase.

Research has been going into the creation of a high COP air conditioning system. This involves operating at optimal pressures and temperatures, better designed heat exchangers and compressors, the development of expanders and the study of more advanced refrigeration cycles involving flash tanks and multi-stage compression.

However one less researched area in this aspect is the air distribution itself. Building designer and researchers have been studying the use of more advanced air distribution techniques to improve comfort and save energy. The latest progress is in the field to personalized ventilations, the use of devices and methods to bring the air or more precisely, the air inlet, nearer to the occupant. Aircraft cabins have been studied to incorporate such methods (T. Zhang & Chen, 2007).

As there has been little or no change in the air distribution in the vehicle ever since the air conditioning was introduced to the car, it would be interesting to bring PV technologies and concepts to the vehicle and study how PV can improve the air flow, bringing energy savings and comfort to the passenger.

1.2 Objective

The objective of the project is to introduce a new air distribution method to the vehicle which has superior performance to the existing vehicle air distribution. Superior performance would mean increased comfort, higher energy savings, better air quality and reduction in dispersion of contaminants.

The aim new air distribution is to minimize the cross contamination of air and allow the occupant to have better control of the air conditions. If possible the air distribution should be practical. This means the design can be used on an existing automotive air conditioning system.

The air distribution is required to attain a pre-defined level of thermal comfort in various conditions and provide sufficient energy savings. As the air distribution is meant for an electric vehicle, energy saving and efficiency would be a priority and should not be compromised for comfort.

1.3 Scope

The project will look into designing a new type of air distribution that can be used in a vehicle, one that uses a personalised ventilation method. These different air distributions would then be analysed to understand the performance of the air flow and to ensure that it can meet the objectives stated above and certain specifications.

In any air distribution project, the first and foremost task is to understand the heat loads coming into the space and the heat sources involved. This is followed by the understanding of the heat load patterns in the space. It is especially important in a vehicle as it is exposed to extreme heat from the sun and has large temperature changes in the car, going from a cool environment during driving and an extremely hot climate when parking.

In this part of the project, the heat load of the vehicle is to be determined under certain solar conditions. This was done in two ways, first via a program written in FORTRAN, next with the use of a commercial package known as Theseus FE. The commercial package is used as a comparison with the FORTRAN programme to see how much the results differed. However, the commercial package offered more versatility and flexibility to the models made and has a more comprehensive thermal balance model.

The results of the FORTRAN model would not be covered in the report. In the report, only the theory and the results from Theseus FE model will be covered.

The scopes of analyses of the vehicle heat loading done via simulation are:

1. Understanding the changing of the heat load profile throughout the day
2. The different phases in the vehicle heat loads
3. How various methods can help in the heat load control of the vehicle
4. How the change in the vehicle geometry can affect the heat load on the vehicle

The study on the heat loading will serve as a precursor to the development and analysis of the vehicle air distributions and also part of the development work for an electric vehicle under TUM Create. It will serve to provide a good understanding of an ideally thermally reduced car.

This will be followed by the design of the air distribution system. CFD simulations studying the concentration of contaminants, humidity, temperature, velocity and comfort would be done. These simulations will analyse different air distribution methods and the results compared. A mesh independence test and verification of the numerical models used would be required to ensure reasonable accuracy.

The project would not be studying in detail the air conditioning system itself but the air delivery methods into the cabin space. This will involve determining the potential issues of

having a PV system, determining the inlets locations, and if necessary explore new devices that may aid in this objective. This PV should be effective even at high solar radiation loading.

The CFD simulations also cover ventilation devices that might have potential for use in a PV system and in the creation of microclimates.

2 Literature review

In this section, the current literature on personalized ventilation technologies will be discussed. This includes methods for channelling the air towards the person, how these methods might work with the overall ventilation system, the physics of the air flows and why certain methods are preferred. A short section on the thermal comfort models used in personalized ventilation will be covered.

2.1 Current air distribution methods

Although the scope of the project is for automotive vehicles, building air distribution methods will also be covered. This is because research into building air distributions has been well studied and a number air flow strategies have been developed for various purposes.

2.1.1 Automotive air distribution method

Automotive air distribution has remained similar ever since the air conditioning system was introduced into the vehicle. The inlets of the air conditioning system are usually found at the dashboard. In modern vehicles, depending on the make and class of the vehicle, luxury vehicles usually have the inlets located in the passenger area too. This gives more control of the air flow into the zones of the passengers and driver.

The temperature in the vehicle cabin is achieved by blowing cold air into the space to bring the temperature of the space down to provide a comfortable temperature to the person. The velocities in the cabin are usually quite low (estimated to be around 0.1-0.2 m/s) and thus the main mechanism for convection to take place is the temperature difference. Similar to conventional room air conditioning, this involves cooling the entire air space down to the required comfort level.

The main issue from such a method is that cooling the whole space to a low temperature is not energy efficient. This might not be a big issue for petrol vehicles but for an electric vehicle with limited battery capacity, this would affect the power available to the vehicle for driving and decrease the range.

Figure 2-1: Typical vehicle airflows

In general, the air flow in the vehicle would be characterised between displacement and mixing air flows. Air is forced through the inlets into the cabin from the dashboard. This mixes with the existing air in the cabin bringing down the temperature to a comfortable level. The air will usually be removed from the cabin through an outlet at the bottom of the dashboard or the back of vehicle. In short, as stated earlier, the heat loss from the body by convection would mainly be driven by the temperature difference of the cabin air and body temperature.

The general advantages of such a system would be:

- Easy to install – generally, the ducting will be minimised with a few inlets and outlets
- Ducting is smaller as flow rates required are lower
- Simple zoning is easy to achieve – small zones with small temperatures differences can be achieved by having the inlets around the person zone to release air at the required temperature. This will cool the respective zone to the required temperature.

However it also has some disadvantages:

- Not as energy efficient – the entire cabin space will be cooled down
- Passengers exposed to heat sources– due to solar irradiation entering the space, the side exposed to the sun will tend to feel hotter. This despite some control strategies in vehicle air conditioning used to direct more air to the hotter side, there will be limitations where this type of system can do. A directed cooling method would achieve better results as air of the right temperature can be focused on the right area.
- Air from the inlet might be too cold and strong – there can be complains from passengers at the back that the air conditioning is not strong enough or that they are feeling warm. But the air conditioning was lowered as the occupants at the front feels too cold. The inlet position in current vehicle (especially if there are only inlets at the dashboard area) will tend to create a colder environment at the front. Moreover, only the middle passenger at the back feels cooler as the middle inlets on the dashboard will blow directly towards him.
- Fresh air is important to the driver but it will require more energy from the air conditioning – In the most of the current design now, fresh air is channelled in for the whole cabin. There is a disadvantage of such air distribution. A directed approach again would be able to release fresh air only to the breathing space of the person while the rest of the cabin uses re-circulated air for energy saving

The advancement in air conditioning technology for automotive vehicles, has been in both better and more efficient air conditioning systems and components, and better control of compressor performance for better energy efficiency. Control systems have been developed by car manufacturers to perform advanced thermal management functions such as sensing of cabin conditions and providing better response with intelligent control of humidity and temperature to increase comfort. One example of this is the system developed by BMW being used in the 2012 3-series vehicles (Ingram, 2012)

Air conditioning is not only important for maintaining a comfortable cool environment and with an appropriate humidity; it also needs to provide fresh air which is higher in oxygen content. Efficient air distribution would assist to ensure that the air in the cabin has fewer stagnated areas and recirculated zones. The build-up of contaminants in the space, such as volatile organic compounds like benzene, formaldehyde and so forth, may be a health risk at higher concentrations. This is a health concern as the vehicle cabin tends to have these chemicals at higher concentrations, for some chemicals, even more than two times as compared to residential indoor air (Geiss, Tirendi, Barrero-Moreno, & Kotzias, 2009). In general, the concentration of these chemicals is always higher in the cabin compared to the outside. Therefore, the source of the volatile organic compounds (VOC) would most probably

be emitted from the materials in the cabin or leakage from the engine etc. The air distribution has to be effective to ensure the build-up of these chemicals is minimized.

In Singapore or in urban areas with high traffic volume, the air conditioning system is usually run using recirculated (RC) air due to the pollutants found on roads and congested areas. It has been found that having the air conditioning in fresh air mode or when the windows are open resulted in higher concentration of pollutants in the car (Chan, 2003). However, there is an increase in risk of spread of airborne diseases such as influenza when the air conditioning is run in RC mode, from 59% – 99% increase in risk (Knibbs, Morawska, & Bell, 2012). Although there is not much literature on how RC mode affects the occupants in the car, spending extended amount of time in the vehicle might result in a similar condition found in buildings with poor air quality i.e. little fresh outdoor air (Wargocki, Wyon, Sundell, Clausen, & Fanger, 2000). This condition is known as sick building syndrome and comes with symptoms such as dizziness, headaches, sleepiness and so forth. This may also occur for vehicle drivers who are exposed to poor quality air which may arise from the use of RC mode for extended periods of time.

2.1.2 Conventional Building air distribution strategies

Ventilation and air-distribution studies in buildings are more established and well researched. These technologies and approaches will be discussed in this section and the knowledge and lessons gained can be applied to enhance and adapt to ventilation and air-distribution in vehicles. Buildings these days are always equipped with some form of air conditioning and ventilation system. As rooms in large buildings are usually not naturally ventilated due to various reasons; ie high environmental ambient temperatures and humidity, difficult to channel natural air flow, lack of windows etc, mechanical ventilation is very important to achieve a comfortable living environment in the living space. Besides maintaining the temperatures and humidity of the space, air has to be exchanged and contaminants concentration kept low. Maintaining good health conditions are also an important aspect of ventilation.

Conventional air distribution strategies (non-personalized ventilation) for cooling the airspace have been developed. Different strategies can cater for different scenarios but with certain limitations. Using the naming convention classification by the (Hagstrom, Sandberg, H.Koskela, & Hautalampi, 2000), the conventional air distribution methods will be discussed below.

Generally, the air distribution strategy can be divided into 4 main types. However in practice, the air distribution might involve a mix of the few these main types. The 4 main types are:

- Piston – This involves unidirectional air flow over the room
- Stratification – Air flow is supported by density differences between the warmer and colder regions resulting in a flow due to buoyancy. This results in the warmer air moving upwards and to the outlet.
- Zoning – Selected zones' air flow will be controlled by the supply air and the other zones will be by stratification or buoyancy
- Mixing – To provide uniform conditions with little or no stratifications

Many of the offices and homes air conditioning distributions now use a mixing type of air distribution strategy. This attempts to create a constant and uniform temperature and humidity in the space. Zoning in turn is quite close to having personalized ventilation with the supply air controlling the air space air flow. The figure below shows the different airflow strategies, from left to right: piston, stratification, zoning and mixing

Figure 2-2: The different air distribution methods, extracted from (Hagstrom et al., 2000)

The table below explains the advantage and disadvantage of each strategy as with some inputs by (Hagstrom et al., 2000):

Method	Advantages	Disadvantages
Piston	<ul style="list-style-type: none"> The airflow is highly controlled allowing almost total air change Areas upstream of the contaminants sources are kept clean as there is little air mixing Good control of temperature and contaminant removal Since flow rate is high, temperature of inlet air might not need to be as low to achieve the required temperature in zone 	<ul style="list-style-type: none"> Will need very high air flow rates to achieve high air exchange rates Large supply inlets and outlet will be needed.
Stratification (displacement)	<ul style="list-style-type: none"> Low concentration of contaminants in ventilated zones as buoyancy is relatively effective to channel airflow towards outlet Good temperature distribution, lower temperatures near the floor, higher near the ceiling 	<ul style="list-style-type: none"> Easily affected by disturbances such as high velocity airflow Stagnant areas with little or no air change is possible
Zoning	<ul style="list-style-type: none"> Contaminant removal and thermal effectiveness better than mixing air strategy Allows for some control of air flow patterns in the ventilated zones Stagnation areas are minimized in the ventilated zones 	<ul style="list-style-type: none"> Inevitably, contaminants from outside the ventilated zone would still be able to mix with the ventilated zone through the air flow patterns
Mixing	<ul style="list-style-type: none"> Low stagnation areas with high concentrations, contaminants are mixed with cleaner region to form a dilute region Little thermal stratifications 	<ul style="list-style-type: none"> High air velocity might be needed to allow for good mixing of air and this high velocity might cause draughts Low overall contamination removal

Table 1: Advantages and disadvantages of different airflow strategies

In order to understand which strategy to use, one would have to evaluate the requirements needed and the limitations of the space or cost. Another important factor would also be the energy requirements. Most office and living spaces makes use of the mixing type of air distribution. However, more spaces especially offices are moving towards a displacement or stratified type of air flow.

One type of displacement ventilation, underflow ventilation systems have the air supplied from vents under the floor or on the floor level. The air is distributed around the space and returns via outlet at the ceiling. Therefore, the general air flow will be often from bottom up, supporting the natural buoyancy of the air. The advantages of such a system are numerous which is why such systems are becoming more popular.

Traditional ventilation systems (mixing type usually) will have the vents positioned at the ceiling, both inlet and outlet. How the air will flow will generally be due to the placement of these vents. Using simulation to analyse the thermal sensation, air movement and temperatures, underfloor ventilation systems were able to provide a superior performance in the living space(Ho, Rosario, & Rahman, 2011).

Firstly, air quality near the occupant is higher as the inlet channels clean air to the occupants. This is due to the floor to ceiling air flow pattern removing contaminants and preventing the mixing of these pollutants. Energy savings of about 20-30% was attainable with the use of underfloor ventilation systems. The thermal comfort results between the 2 systems were very close. However, probably due to the inlet placement on the floor and the outlet, cooler temperature was found a distance away from the user. Cooler air should ideally be push into the person's airzone and warmer temperature for areas that are not cooled. This can be studied in order to optimize achieving better comfort and energy savings. However, this shows that underfloor ventilation is able to offer more control of the regions to be cooled. Other studies via experiments also showed that the age of air in the space was improved with an underfloor ventilation method (Chao & Wan, 2004).

For a vehicle, the current strategy uses mixing with some features of piston. The air from the inlet is pushed in and mixes with the cabin air. However since the cabin is quite small, the air can also be considered to be forcefully pushed from the inlet to the exit. To understand which air distribution method would be ideal for the vehicle, it would be important to identify the properties of the vehicle space and environmental along with the goals that the vehicle air distribution should achieve.

Building air distribution research is moving to personalized ventilation methods. With this, research is also being done to better understand the type of air distribution methods which might assist in the usage of personalized ventilation devices. This will be covered in the next section.

2.2 Personalised ventilation

The more recent and significant developments in this area will be covered. The results of the experiments and simulations will be presented along with the knowledge or lessons that can be applied to the vehicle. In this section, PV technologies and methods from buildings will be

covered as they are more established. This will be followed by aircraft and vehicle PV methods that had been researched.

Personalised ventilation (PV) has gathered interest in the last few years as part of the air distribution strategy development. Although most of the focus in the past has been in the area of offices and workspaces, an important area which has gathered interest is in hospitals. This helps to reduce contamination and spread of diseases in wards (Arsen K. Melikov, 2011).

It has been shown that personalized ventilation has the potential to improve the conditions in the living space. For studies concerning energy usage, PV has been able to reduce energy usage significantly. An example of this would be the use of recirculated air only for cooling of the office space while fresh/outside air is only pumped in through the PV inlets. This allows for energy savings and a higher quality of inhaled air. Results have indicated that subjects do prefer a PV system to a mixing type (Sekhar, 2007).

HVAC research is moving towards PV based systems as there is a demand for such systems. Personal preference can vary greatly due to various differences in individuals, from their dressing, activities they are engaged in, personal preferences, metabolic rate and so forth. It was found that personal preference of air temperature can vary up to 10° C and air velocity can vary up to 4 times among individuals (Grivel & Candas, 1991).

PV being able to push a higher percentage of fresh air towards the user's breathing zone, allows for 2 benefits, namely higher percentage of fresh air inhalation and lower inhalation of contaminated air. This resulted in better health benefits and less risk in a condition known as sick building syndrome (SBS). Studies have been made to compare the performances of PV and mixing ventilation, evaluated with the help of human subjects (Kaczmarczyk, 2003; J. Yang, Kaczmarczyk, Melikov, & Fanger, 2003). It was found that the PV system was preferred as the air from the PV was perceived to be of better quality and lower SBS symptoms were observed. In fact if the air from the PV system was cooled, even better results were obtained.

Another important area which PV systems were studied was how air from PV system and the ambient temperature affected the perceived comfort. It was found that isothermal PV systems were greatly preferred as the temperature in the air space increased above 23°C. However above 26°C, if the PV system used only cools a small area of the body, the cooling effect might not be sufficient to achieve good comfort and if velocities are too high, draught discomfort would result. It was also found that people preferred flow velocity were from 0.2 m/s to 1.2 m/s (A. K. Melikov, 2004). A number of papers also stressed that having personal control of the PV system would allow people to achieve greater comfort with using this system and lower the risk of draught discomfort. Research has also shown that for an ambient temperature of about 23-26°C, air from the PV at about 20°C was deemed comfortable (Kaczmarczyk, Melikov, & Fanger, 2004).

Studies have shown that PV systems used, in conjunction with mixing ventilation (MV) or under floor ventilation systems (UFV), have better ability to reduce the spread of contaminants like pollutants and diseases when comparing PV systems with MV and UFV. However this might lead to higher risk of transmission to users who are not using the PV systems. This is a similar disadvantage to the zoning air distribution strategy discussed in the

earlier section. PV systems will tend to push the contaminants away and towards the uncontrolled zones. This is a design issue which has to be studied.

Another major issue about PV systems is the increase in ducting for the inlets and outlets. The primary purpose of PV systems is to channel air flow to where it is required. It leads to the need to ensure that the air flow patterns within the overall space are suitable and optimized to ensure good living conditions. The increase in ducting can result in installation difficulties and complexities in vents and valves controls which usually not covered in the scope of these research studies.

2.2.1 Building personalized ventilation methods

In this section, the research on building personalized ventilation methods will be covered, most importantly, information that can be applied towards the automotive air distribution. This includes significant and more recent literature on PV in buildings and the result of such analysis.

Building PV methods have been targeted towards office workplaces as the positive effects of PV would be best appreciated in such a setting. Office workers may stay at their workstations for hours at a time and remain in the office environment for the entire day. Therefore, having a properly ventilated area is important for health, comfort and productivity. Office spaces are also the easiest for the installation and usage of such devices as the spaces do not change often and the office workers do not have large movements or shifting of positions.

Different ATD devices

A variety of PV air terminal devices (ATD) have been designed and tested by a number of researchers. ATD designs vary in shape, size and position, each resulting in different performance in terms in airflow, fresh air percentage and local comfort. A comparison of the common type of ATD inlets for PV system was made comparing 5 types, namely: moveable panel (MP), computer monitor panel (CMP), vertical desk grill (VDG), horizontal desk grill (HDG) and personal environments module (PEM). A manikin which is able to simulate breathing and body temperature, it was placed in an office environment to test the performance of these 5 devices.

Figure 2-3: The five different type of PV ATD, extracted from (Arsen K Melikov, Cermak, & Majer, 2002)

The ATDs were evaluated using an index known as personal exposure effectiveness which serves as a measure of the amount of contaminants in the air. This is an improved form of the ventilation effectiveness. Other measure of effectiveness was the re-inhaled exposure index.

Assessing the personal exposure effectiveness of the ATD highlighted a number of important issues (Arsen K Melikov et al., 2002). Firstly, all ATDs require a minimum amount of flow rate in order to attain a certain amount of effectiveness in terms of fresh air inhalation. This can be attributed to the fact that when the air is able to reach the user, only then would it be considered effective. Ideally the air should be able to penetrate the free convection thermal plume in order to be able to bring a good amount of fresh air to the inhalation zone. This will depend a lot on the air flow rate from the ATD, the distance the ATD is away from the user and the cross sectional area of the inlet which affects the velocity. Secondly, it was shown that VDG was able to give the best performance attaining the highest personal exposure effectiveness among all the ATDs and it did it with the lowest flow rate. MP was a distance second place at lower flow rates.

The effectiveness of re-inhaled exposure is actually very low for all the ATD with a less than 1% of exhaled air being inhaled. VDG performed the best. Interestingly, it was found that CMP and MP resulted in the increase of re-inhaled air with the increase of the flow rate. This can be due to the warmer exhaled air being pushed back into the inhalation zone as the ATD blows air from the top.

The temperature of the air inhaled is also an important factor to consider giving information on how much external ambient air has mixed with the PV air stream. It was seen that VDG again was the best performing giving a drop of 5°C. MP and PEM also performed well in this case, providing a drop of 3°C.

Assessing the ability of the PV to provide effective heat removal, comparing all the ATDs at the flow rate which provides the best personal exposure effectiveness, MP performed the best but with quite a high flow rate. However, from the results it can be predicted that at the flow rate at 10 l/s, which MP performed only marginally better than the rest for personal exposure effectiveness, the heat removal would be slightly better than the rest except VDG.

However more research should be done in terms of thermal comfort of these devices. As PV tested would push air towards the front of the body resulting in less discomfort compared to the back, the high flow rates it was tested at might be an issue in reality. It can be concluded that the 2 ATDs VDG and MP has the best ability to provide PV performance based on the criteria it was measured against. Such PV locations can be evaluated for use in a car.

Another study similar to the above was conducted, this time with the use of human subjects to understand their issues and the effectiveness of 5 different ATDs (Kaczmarczyk, Melikov, Bolashikov, Nikolaev, & Fanger, 2006). The ATDs are different from the study mentioned above however. Figure 2-4 shows the 5 ATD tested.

Figure 2-4: 5 different types of ATD tested with human subjects, extracted from(Kaczmarczyk et al., 2006)

It was found that the about 50% of the subjects were satisfied with the round moveable panel (RMP) +HDG or MP and 70% preferred the RMP. The common thing feature of these setups was the presence of an inlet at the face or head level. Interestingly, even though RMP and MP have similar designs, RMP was preferred. This can be attributed to the design of the RMP. RMP has a larger surface area which allowed for lower velocity area and care was taken during design to ensure the air output had less turbulence. This resulted in less mixing of the clean air with the ambient air. Note that for all the tests, the temperature of all the ATDs is the same, fixed at 20°C. Only the ambient temperature is changed. The results also showed that in general, most people do not like getting cold air at the abdomen. The RMP + HDG ATD had the HDG switched off about 50% of the time regardless of the temperature or pollution level. As a reference, the subjects used a flow rate of about 11-13 l/s in most of the cases.

Another type of PV is the ceiling mounted type. It is quite different from the usual close proximity type. This PV is of interest by researchers for its ability to be installed easily with the existing air conditioning ducts, reducing the need for air channels though the floors and the to the desk as desk inlets. However, as it is mounted on the ceiling, such PV would need to have flow rate sufficiently high in order to reach the user as they would be situated a distance away (1.3m).

The experiment studied the scenario when the PV system was used with MV air distribution. It was not effective in providing thermal cooling of the body parts (B. Yang & Sekhar, 2009). The lowest flow rate of 4 l/s only provided a drop in equivalent temperature for the body segments of 1°C and a drop of 6°C when a high flow rate of 16 l/s was used. In fact only certain regions of the body exposed to the ceiling nozzles were cooled despite higher flow rates. Ceiling PV system can be considered most adaptable to user movements as the cooling region from the ceiling would have a larger span.

Although ceiling PV would have the ease of installation and usage, due to the distance from the person, very high flow rates are needed to provide sufficient cooling. Even so, some region of the body might not be sufficiently cool. Therefore, ceiling PV can only be used to

provide some improvement to the MV and not as a compliment to existing air distribution system. If possible a PV system which is mounted closer would be more ideal.

Figure 2-5: Flexible PV allowing adjustment of position, extracted from (Sekhar et al., 2005)

A study conducted in Singapore researched into the area of using flexible PV. Flexible PV allows the user to adjust the PV inlet to any position they prefer. It was observed that most of subjects placed the ATD very close to them, directing the inlets towards their inhalation zone. What is interesting is that they were comfortable with the PV's position for an extended period of time regardless of the flow rates tested. In fact for the ambient temperature of 26°C, cool air at 20 -23°C and a flow rate of 7 to 15 l/s, the thermal comfort rating given by the subject increased as the flow rate increased and temperature dropped (Sekhar et al., 2005).

Depending on the ambient temperature and the body temperature, especially when the temperature difference is large, the free convection air flow or thermal plume of the person can be very strong. This will result in the required flow rate or velocity needed for the PV jet to be high in order to penetrate the plume. In such cases, resorting to strategies to weaken the plume might be ideal instead of increasing the PV velocity. One simple strategy which was found useful is to allow a small amount of cold air from the PV to flow towards the chest area (Z. Bolashikov, Melikov, & Krenek, 2010). This would cool and weaken the thermal plume making the resultant flow at the inhalation area weaker.

Other concepts include a combination of PV devices to create a individually controlled microenvironment (A K Melikov & Knudsen, 2007). This involved the use of a number of devices shown in the Figure 2-6.

Figure 2-6: Individually Controlled Microenvironment, extracted from (A K Melikov & Knudsen, 2007)

The PV systems mentioned above, all requires ducting from the air supply to channel air to the user. A ductless type of PV was studied (Halvoňová & Melikov, 2010; Arsen K Melikov, 2010). This PV will obtain the supply air from the ambient air similar to a fan. The only issue is that this type of PV needs to be used only in a space that is cooled by displacement ventilation. This means that the supply air will be supplied from the bottom (floor level) and the air is exhausted at the ceiling. The advantage will mean that no extra ducting needs to be installed just for the PV. However, some concerns will be the quality of air as the air would be drawn from the floor area. Nevertheless, the air quality and comfort was found to be enhanced as the air drawn was closed to the supply, resulting in the supplied air by the PV still being cooler and cleaner than the ambient air around the body of the user. This PV similar to others was still able to perform the essential task of blowing air targeting a particular part of the body such as the face and the inhalation zone increasing the perceived air quality.

Assisted PV

One idea that might arise would be the use of fans to aid in the convection cooling process of the body. An experiment involves the use of the PEM type of PV positioned on the desk facing the user, with a desk mounted fan (DMF) found under the desk. A CFD simulation was used to carry out the different variations in parameters.

Figure 2-7: PV ATD with desk mounted fans, extracted from (Tham & Pantelic, 2010)

The results showed that having a desktop fan will generate increased convective cooling in the abdominal region due to the increase in velocity (Tham & Pantelic, 2010). The personal

exposure effectiveness was also increased due to the reduction of the convective layer over the manikin. When the PV inlet was positioned quite close to the user, it created a boundary which the DMF air stream was not able to penetrate into the inhalation zone. DMF also serves the purpose of preventing pollutants from entering the inhalation zone by the reduction of the convective boundary layer due to the thermal plume. One major advantage of such a coupling is that it helps to distribute the cooling over a larger area and thus reducing the risk of draft.

Another idea is to make use of fans to redirect the flow in a certain direction to control the flow of the air around the body, this helps to reduce the strength of the convective plume from the body (Z. D. Bolashikov & Melikov, 2009). The use of passive control methods such as body contour fitting tables tops can also be used to control air flow going up the body. Such simple methods were able to improve the personal exposure effectiveness by a significant amount at lower ATD flow rates. This is due to the ATDs being positioned near the breathing zone requiring less flow rate to penetrate the convective plume. In fact, with these enhancements, a maximum improvement with the personal exposure effectiveness (PEE) was about 190%.

Energy efficiency

Studies have been made in the area of studying how PV systems can improve the overall energy consumption required by the HVAC systems.

It is shown that PV does indeed have the lowest energy consumption. A study, (B. Yang, Sekhar, & Melikov, 2010) compared the PV system with a MV system, showed that the PV system offers about 2 to 4% improvement. In fact it can be seen that the ducting and the channelling of the flow towards the PV nozzles was large, reducing the advantage of cooling a smaller load. However, this is a design issue for the ducting and can be improved. In a vehicle this might not be a significant issue as the PV system would most probably use similar ducting to what is used in the vehicle right now.

Another study done by a different group however showed that improvements in using a PV system can result a 51% savings in energy consumption (Schiavon, Melikov, & Sekhar, 2010). The strategy used by the study considered the reduction in airflow rate due to a more effective cooling due to PV. Moreover, the cooling of the occupant space will only occur when the user is present taking into consideration occupancy rates.

The difference in results can be attributed to the different assumptions made of the usage of the HVAC system. The 2nd study focuses on the room usage taking into consideration occupancy rates for the week and the use of some energy saving strategies. The former study adds the influence of ducting and channelling losses.

2.2.2 Aircraft Personalised Ventilation Methods

There are a variety of aircraft PV methods being proposed. These methods serve to improve the air distribution in the cabin in terms of comfort and contamination. Aircraft HVAC system uses the outside air as a source of fresh air to mix with the recirculated air as a source of oxygen and to reduce the humidity in the air. The outside air is extremely dry at high

attitudes and allow for the humidity in the cabin to drop to about 10%. The objective of this is to minimise the risk of dew condensation. In conclusion, the air would be required to supply sufficient oxygen, pressurize the cabin and to dilute the contaminants concentrations.

One scheme proposes that the inlets be placed on the armrests of the seats (T. (Tim) Zhang, Li, & Wang, 2012). This system works with an under floor ventilation system. The cabin air inlets are placed in the aisles with the outlet situated at the top of the ceiling. With the armrest air inlets positioned in a cross fashion facing upwards, the general airflow pattern will be from the lower cabin levels towards the ceiling where the air will be exhausted. Therefore, the buoyancy of the air due to heat sources resulting in the thermal plume will be in the same direction as the cabin airflow. This allows for low velocities in the cabin which would not be disrupted by the heat sources inside the cabin. The inlets for the personalised ventilation blow air upwards from the armrest towards the face and chest. This brings fresh, clean air to the passenger with low contaminant content.

Figure 2-8: PV inlets at the armrest area, extracted from (T. (Tim) Zhang et al., 2012)

The experiment was carried out with only one person equipped with the PV system and was used only to validate the CFD simulation. 2 CFD cases apart from the validation case were run, one with the PV system off only the MV system functioning and the other with all the passengers equipped with the PV system with a flow rate of 4.5 l/s per person. One issue with the fairness of the case, the case 1 and case 2 should have the same overall flow rate. Instead the MV system has the same flow rate for both cases of 0.264 m³/s. The PV system will feed an additional 4.5 l/s per passenger.

Figure 2-9: Armrest PV inlets cabin airflow, extracted from (T. (Tim) Zhang et al., 2012)

The results obtained were positive. This method works by blowing air out allowing the 2 air streams to collide with each other in the inhaling region of the passenger. The velocity of the supply air is about 0.65 m/s and it benefits from the thermal plume of the passenger. In terms of ventilation efficiency, such a system can reduce the pollutants in the breathing region by 40%. The pollutants are concentrated overhead of the passenger closer to the outlets. It was found that as the supply with the inlet air at 24.5°C is located close to the passenger, the region near the passenger can be maintained at a temperature close to the supply air of about 25°C. One concern of a under floor ventilation system is the temperature stratification. However the author concluded that the temperature gradient of only 2°C /m was acceptable. Using the thermal comfort/percentage dissatisfaction criteria from the ASHRAE handbook was found to be about 8% for the region around the passenger.

Although the performance of such a scheme is quite good, in a vehicle, this might not be ideal. Armrests might not be available for a vehicle. If this air flow scheme is to be used, some modifications have to be applied. However, having an inlet source near the user would give numerous advantages as stated above.

The next setup proposes the addition of a nozzle jet at the front passenger, mounted to the rear of the passenger seated in front (T. Zhang & Chen, 2007). This adds an additional personalised air source near to the passenger breathing area, providing a fresh source of air for the passenger. The PV system is to be used with displacement air distribution.

Figure 2-10: Personalised inlet air flow, extracted from (T. Zhang & Chen, 2007)

Zhang compares this PV system with 2 other types of air distribution methods , namely:

- Mixing air distribution
- Displacement air distribution

Figure 2-11: The convectional mixing ventilation for aircraft cabins, extracted from (T. Zhang & Chen, 2007)

The mixing air distribution has inlets placed around the ceiling area while the outlets are found on the sides of the cabin floor next to the wall. The displacement air distribution has the air inlet on the floor, next to the aisle. The outlets are found on the ceiling. This creates a scenario where majority of the air flow is driven in a one direction from the ground upwards to the ceiling. However, some mixing can occur at regions a distance away from the inlets.

CFD simulations were done to observe how the contaminants will spread due to the different air distributions. The velocity, temperature and CO₂ levels for certain points in the cabin were compared for the 3 distribution types. From the results, it showed that the mixing air distribution tend to have a higher velocity especially at the ceiling area. At the level closer to the passenger away from the walls, the velocity was comparable. The displacement air distribution had the lowest air velocity as the surface area of the inlets into the cabin was large and main driving force was from the thermal plume generated from the passengers. The highest velocity near to the passengers was generated by the personalised system. This is required as sufficient velocity is needed to penetrate the thermal plume boundary layer to cool the body. However, the velocity near the body was lower than 0.25 m/s and therefore, should not be felt by the passenger.

The temperature of cabin showed that the mixed distribution had the most uniformed temperature in the cabin. This implied that air was well mixed. For the other 2 distributions, temperature stratifications exist. However, the author concludes that small temperature difference between the head and ankle should not give any discomfort as it was lower than 3K.

The carbon dioxide concentration results also showed that mixing air distribution resulted in a uniform concentration around the cabin. The carbon dioxide introduced into the computational space came from a point source on the passengers. The personalised distribution resulted in the air near the floor having higher CO₂ concentration and the

breathing zone of the passengers having a lower concentration of CO₂. This is due to the air inlet on the aisle supplying only re-circulated air and the personal nozzle supplying the fresh air.

The air stratification allowed for lower mixing of air, leading to improve air quality at the passenger's breathing zone. Having air mixing strategies would not be ideal. However having an underfloor inlet air distribution system should be avoided if possible due to the simple fact that dust and smells from the floor area might be blown upwards to the breathing zone. Moreover the inlets if not properly designed, might be prone to blockage by bags or items that are placed on the floor.

Another version with some changes has been made to Zhang's method of the cabin PV air distribution. The concept proposed by Zitek et al seeks to combine the personalised air flow field with the cabin mixing arrangement flow field (P.Zitek, Vyhlidal, Simeunovic, Novakova, & Cizek, 2010). The passengers will receive air at 2-3 l/s at a humidity of 40% while the main air distribution supplies air at 6.5 – 9 l/s at a lower humidity. Each passenger has 2 nozzles, one air inlet and the other is an air outlet. This will create the continuous flow of air being maintained at the passenger breathing area. It was shown that for such a system, one will need the inlet to have a temperature lower than 2 – 5°C lower than the outlet to maintain closed flow pattern airflow. However for most HVAC conditions, this is not an issue.

For the first part of the study, an experiment was done to ensure that some form of close air flow field was achievable for the required distances and speeds in the cabin for one person. In this part of the simulation, the influence of the cabin ventilation system is not considered.

Figure 2-12: PV ATD with mixing air distribution, extracted from(P.Zitek et al., 2010)

Particle image velocimetry methods were used to observe how the air flowed and the FLUENT simulation was used to verify it. From the flow patterns, it can be seen that for such air flow designs, the flow pattern was only achievable when the temperature difference of the inlet is maintained at least 2-5°C below the outlet. When the temperature different is not maintained ie, inlet becomes warm, the closed air flow field will not occur.

Figure 2-13: Temperature difference affecting the PV airflow pattern

The results for the full cabin simulation showed that in terms of maintaining a close flow field for each person, this method of air distribution control is possible. From the results, it can be seen that for people sitting side by side each other, the humidity for each person was still very distinct. However the CFD simulations should take into consideration concentration of the particles or air that has moved from one person to the other.

A design by Gao et al uses, an ATD with a flexible gooseneck hose which allows the fresh air to be brought very close to the face (Naiping & Jianlei, 2008). The cabin ventilation used is the MV scheme. The results were positive having a person's dissatisfied percentage of only about 15% with a flow rate of 2.5l/s or a velocity of 0.5 m/s with the temperature warmer than the ambient. However, such PV system might be impractical in reality compared to the other methods stated above.

2.2.3 Vehicle personalized ventilation methods

Developments into vehicle PV systems have only begun in recent years. Another vehicle development has resulting in vast improvement in terms of efficiency of engines, control systems, safety devices and so forth, vehicle air distribution and conditioning have not seen much significant progress. This could be due to the fact that vehicle air conditioning is considered an auxiliary in a vehicle. As discussed in the earlier sections, with the drive for energy efficient systems and the attempt to improve the comfort in vehicles, PV systems in vehicles have started to gain interest. Research in aircraft PV systems is gaining momentum, but vehicle PV systems are still in its infancy.

A few research projects have been embarked on in this area. One idea was to create a microclimate with the use of inlets positioned at the side and top of the cabin area. This serves to create a vehicle cabin that allow for a regional steady state temperature. This allows each passenger will have their own area with their own personal temperature demands. The airflow would be controlled by strategically placed air inlets and outlets in order to adjust the temperature of the space and to create a zone independent of the others.

An experiment was conducted using a Ford Liata chassis with the original interior trimmings. HVAC inlets and outlets were modified (Huang, Tzeng, Jeng, & Chiang, 2006). The airflow to be studied will only be restricted to the small space. The air inlet is placed at the left side roof while the air inlet and outlet is placed on the right lower side at the door. The air outlet will

create a barrier to separate the region between the driver and the passenger. An air temperature sensor is placed under the headrest of seats to detect the temperature of the air. This air temperature in the space and the surroundings would be detected and feedbacked to the control system, and the air speed and temperature will be adjusted accordingly.

Figure 2-14: Air inlet and outlet positions to form microclimate, extracted from (Huang et al., 2006)

Although the experiment is quite limited in the scope, based on the temperatures obtained from the controlled and non-controlled zone, maintaining of the independent zones temperatures is quite effective. The temperature difference between the overall cabin and controlled zone was about 2-6°C. One issue about the results of the study is that only with the temperature of the spaces are given without velocity flow patterns. Velocity and temperature of the air is quite important when the comfort of the passenger is of concern. Even if the temperature of the space can be maintained at the desired value at the expense of high velocity air, this will affect the comfort of the passenger greatly. Moreover, upon observation of the air inlet, one of the air inlets is placed in a position blowing directly to the side of the passenger. This might not be ideal if the velocity is high and the air inlet is cold.

If a PV device core purpose is to perform cooling, the PV system would be able to either create a cooler microclimate or to project high velocity air towards the body. Each type makes use of methods that can enhance convection. The latter method was researched and it involves the placement of nozzles targeting certain areas of the body.

Known as spot cooling, it aims provide cooling via convection using air velocity rather than having cold air (Ghosh et al., 2012). This means that, one can utilise a warmer inlet air temperature to provide the same amount of convective heat transfer from the person, leading to lower energy requirements from the air conditioning.

Figure 2-15: Example of spot cooling, extracted from (Ghosh et al., 2012)

A requirement of this method would be that nozzles need to be positioned close to the body. The air blown out would be of a specific temperature and flow rate for each part of the body to give the required level of comfort. Parts of the body such as the chest and waist will have higher flow rate requirements. Personalised cooling in an automotive is recommended but would be a more complex setup in a vehicle. The main reason is that the thermal loading in a vehicle can change rapidly. In addition, a vehicle has a smaller air space compared to an office space for example, this means that changes in temperature due to the heat load can occur very quickly.

The spot cooling scheme in this case looks of combining this with the conventional HVAC system. The conventional HVAC system would then provide a lower percentage of the total cooling. The 3 scheme of cooling were tested, face and chest cooling, face and lap cooling and lastly, face, chest and lap cooling.

Face cooling aims to cool the face of the person with the use of a low airflow air stream. This will prevent the people from feeling a draft. However, the actually velocity used in the simulation is not known; only the volumetric flow rate is given. The placement of the nozzles was explored. For the face cooling nozzles, these can be placed at the headliner (roof) area or the rear of the seat of the passenger at the front.

Chest cooling will aims to have air at higher flow rate blown to the chest and abdominal area. If directed correctly, this might allow the armpits of the passenger and the sweat in that area to be evaporated. In fact the chest cooling nozzles are the most important for comfort they will cool a large portion of the body which is very critical for comfort. The nozzle placement can be position at the A, B pillars or the frame of the car. Velocities should ideally be not too high when the temperature is low for air blown towards the abdominal area.

Lap cooling is also found to be very effective as it is able to isolate the people from the warmer cabin. This is done by blowing air over the lap area in which the air would be forced up to the chest area. In turn, this results in the airflow covering up to the chest area. The issues the author found with lap cooling is that the airflow needs to be high if it is to allow the air to be pushed over clothes. Due to the requirement of high velocity airflow, the

temperature of the air for good thermal comfort would be important. If the temperature is too low, coupled with high velocity air, the person will feel too cold.

Seat cooling is another aspect that was explored. Seat cooling looks the design of a seat with a permeable fabric for air on the seat. This allows air to flow out of the seat to cool the part of the person that is in contact with the seat. It is effective as almost 30% of the body is in contact with the seat. Cool air will be pumped directly into the seat cooling both the seat itself and the person.

It was found that to attain the same comfort level for the manikins, spot cooling only required air at 95 cfm at 17°C to maintain the cabin at 29°C equivalent homogenous temperature (EHT). This is compared to the conventional system which needed 105 cfm at 5°C to maintain a cabin temperature of 24°C EHT to attain the same level of thermal comfort.

The main reason for such good performance is that the spot cooling has 60% higher velocities on the major parts of the body. Therefore, as stated earlier, a higher temperature can be used to attain the same amount of heat transfer. This strategy shows the effectiveness of spot cooling. For the goal of a personalised cooling and microclimate, this direction is possible. However, high velocity air might not be beneficial if a microclimate is to be maintained.

Most recently, researchers from Hyundai has looked into modifying the existing air conditioning air distribution to provide zonal cooling (Kwon, Lee, & Foster, 2012). The basic changes were to the HVAC box or the distribution system to allow air to be channelled to the right zones. This would allow only selected zones to be cooled. A new control algorithm was also introduced to assist in the control of such system.

Due to the selective cooling of the individual zones, the unoccupied zones are minimally cooled and the temperatures in these areas are allowed to rise. This would consequently result in energy savings. A reduction of 17% was obtained for the HVAC energy consumption and the range of the EV vehicle increased by 4%. More savings are possible if the air flow distribution was changed. The present air conditioning flow pattern was meant for an entire cabin cooling strategy and therefore, air will be prone to leak to other zones. This results in cooling of other regions too.

Mounting of PV systems in the vehicle is an issue that has to be studied. Such as the air distribution models mentioned above, it involves the placement of nozzles in a variety of areas. This range from the headline regions to the A and B pillars. One position that can be studied would be the mounting of the inlets to the seat itself. One advantage of this would be that it would allow the PV inlet to be brought closer to the person and it allows mounting in any direction.

One concept looks at vents bring mounted on the seat headrest to force air into the breathing zone of the passenger (a. Melikov, Ivanova, & Stefanova, 2012). These inlets would then be able to provide clean air directly to the breathing zone of the user.

Figure 2-16: Seat headrest PV ATD, extracted from(a. Melikov et al., 2012)

It was shown with the use of a manikin, the personal exposure effectiveness (PEE) was the highest at the ambient temperature of 26°C for different air flow rate (3 ambient temperatures were tested, 20°C, 23°C and 26°C). This is because it would be easier for the personalised flow to penetrate the thermal plume generated by the free convection. As the ambient temperature goes higher, the difference between the body and the ambient becomes smaller, lowering the free convection boundary. It was also found that keeping the air supply terminal vertical to the face of the person would result in the best performance for various temperatures and conditions.

The human response on the comfort of such a system was studied with the help of human subjects. Using the same set of ambient and personalised air temperatures, it was found that the difference in the clothing insulation worn by the subjects did not differ much. Basically only half of the subjects felt the air flow generated by the personalised flow. Most importantly, the subjects did not find that this system caused any discomfort and therefore, the draft is not an issue at the conditions tested. As the temperature of the PV goes lower than the ambient temperature, more users felt slightly cold on some parts of the body.

It was found that airflow with a low air velocity of about 0.1-0.13 m/s and a higher flow rate, was effective for penetrating the free convection flow/thermal plume of the human body. This can be achieved using a larger cross section inlet. One important consideration would be the mean background air velocity, in this case about 0.06 m/s and the ambient temperature should not be too low. High background velocity and low ambient temperatures might disrupt the flow from the system. Nevertheless, the use of this method or a personalised air supply would be able to improve the quality of air supplied to the user with little or no discomfort. Higher ambient temperatures with respect to the PV can affect the comfort and air flow, but still within an acceptable tolerance.

2.2.4 ATD Nozzle Design

One major influence of PV ATD devices would be the design of the nozzles itself. A lot of the PV designs presented in the earlier section only looked at the placement and positions of the PV nozzles. The research into the type of nozzle used or the air jet properties are seldom looked into.

One interesting question that arises would be how turbulence of the inlet flow might affect PV performance. Experiments have been conducted on 2 inlets, one high turbulence and the other low turbulence, over a variety of flow rates and temperature combinations (a. Melikov et al., 2012). It was found that low turbulence air was able to maintain a high velocity throughout the entire direct flow path. However for regions away from the flow path, high turbulence inlets will be able to generate higher velocities. The temperature of the air increases as the air gets nearer to the user. Low turbulence ATD was found to result in greater thermal asymmetry of the body and on the same note; it was able to produce higher cooling for the face region which the inlet was directed at. However the turbulence of the ATD did not affect the overall heat transfer of the entire body resulting in similar equivalent temperature for both cases.

The difference in the performance of high turbulence and low turbulence jets is the fact that a high turbulence jet will encourage mixing of the inlet air stream with the ambient air due to the vortexes generated. This mixing will heat up the PV stream if the ambient temperature is higher and its velocity lower. Therefore, having low turbulence air stream from the PV system would be ideal giving more control on the air flow parameters.

There are some nozzles that have been specially made for PV purpose with the aim of supplying high quality air to the user. As explained in the above paragraph, one issue is the entrainment of the clean air source to mix with ambient air due to turbulence at the boundary of the air jet. This results in lower quality air when it reaches the user.

Figure 2-17: Reduced mixing PV nozzle, extracted from(Khalifa, Janos, & Dannenhoffer, 2009)

One solution was to create a nozzle which blows out 2 types of air, the outer rim consisted of ambient air surrounding an inner outlet of clean inlet air (Khalifa et al., 2009). It was found that for a flow rate of 2.4l/s, the ventilation effectiveness of 7 is attained compared to a standard nozzle achieving only about 2. For a standard nozzle to attain the same level of ventilation effectiveness, very high flow rates of clean air will be required. This device would allow for a lower clean air requirements but with the ability to supply clean air to the person.

Figure 2-18: Aspiration annular jet with microclimate, extracted from (F Penot & Meyer, 2000)

Penot et al designed an annular nozzle which is able to create a microclimate air space in the centre of the air flow jet (F Penot & Meyer, 2000). This air flow just would then be separated from the ambient environment. This would be one way personalised ventilation can be maintained within a particular space.

In order to do this, the nozzle has to have 2 separate parts, an inlet and outlet. On the peripheral is the air inlet (air to be released) while the centre of the device is the air outlet (air to be sucked in). Noticed from the figure below, there are 2 flow directions after the air flows out of the inlet. The primary flow spilt from the main air flow and goes towards the outlet. Although flow rate for the device at the inlet and outlet is the same, entrainment of the ambient air allows mass conservation of the air at the inlet and outlet be conserved. This allows the secondary air flow to be possible and it pushes downwards to the area that needs to be cooled.

The experimental results showed that such a device is able to provide an airflow which is suitable for the creation of this microclimate. It can be seen that the mixed convection effects has little effect on the temperature stratification lines.

As for the velocity, one can see that only a small percentage of the inlet air will be suck into the outlet. This shows that such a design is feasible. The advantage of this design would be that it is possible to have an large isothermal temperature separation between the ambient and cooler air in the jet zone, allow for an area where the required comfort conditions to be achieved. This design would allow for some form of microclimates to be achieved without the need of walls or boundaries.

Figure 2-19: Velocity field of aspiration annular jet, extracted from (F Penot & Meyer, 2000)

Penot et al also developed of another type of annular jet with aspiration which generates another type of air flow pattern. Unlike the earlier one which will create a microclimate region at the centre of the jet flow, the newer type of flow aims to create a region of low pressure which is able to create a form of suction. this leads to a high amount of air change (François Penot & Pavlović, 2010).

Penot did an experimental study of the airflow using a similar setup as the initial test. The major difference is that an impingement plate is now added to the setup. Not much information is actually given on how the different parameters might affect the flow or performance.

However from the results shown on the experiment, a small tornado like air flow was created using the flow rate of 0.02 to 0.06 m³/s. The velocities of the inlet and outlet regions were between 0.5 to 1.5 m/s. In this setup, the dimensions of the ventilated zone were about 0.55m diameter with a height of 0.7m. From the visualization results, the presence of an aspirating ascending jet with a swirling flow can be seen.

Figure 2-20: Aspiration annular jet creating a tornado like effect, extracted from (François Penot & Pavlović, 2010)

It is unfortunate that there is not much information on these types of airflow devices. Although there are many papers on the standard annular jet (with only inlets on the jet face), there is no other papers on the annular jet with aspiring centre. This device would be very good for PV applications as it supports the creation of a microclimate.

2.3 Thermal Comfort Models

Thermal comfort is one of the critical considerations when designing or planning for an air conditioning and distribution system. Simple numerical and empirical models have been developed for engineers and designers to gauge if air system design is suitable and sufficiently comfortable for the zone that it is being used in.

Thermal models of a human body can be categorised into a number different types. This range from physiology to psychology types in addition to models which combine the two (Foda, Almesri, Awbi, & Sirén, 2011). Physiology type of thermal models look at how the body behave in a physical sense, how the body reacts to certain temperatures which results in reactions for maintaining heat balance ie, sweating. Such models will give results such as body and skin temperature in response to the ambient temperature and the local heat loads.

Psychological models will look at the perceived sensation or comfort level. Sensation describes how the person will feels towards in a certain environment, if it is warm or cold. Comfort will categorise the sensation, describes the person perception of the space and the level of comfortable. These models are usually empirical models based on experiments. Results would be obtained from human subjects in term of their perceived comfort level and this data would be used to form an empirical relationship with respect to the ambient or body temperatures. Psychological models can be grouped into 2 different categories. One determines the local thermal sensations for a part of the body, the other calculates the overall body thermal sensation.

Combined models combine the physiological and psychological models into one given a complete solution.

Thermal models can also be grouped into physical and numerical models. Physical models in fact life sized physical manikins that are able to generate body temperatures and heat loads similar to a real human body. More advanced manikins can even produce effects like sweating and shivering reactions to mimic such thermal balancing reactions of the body.

In this section, more widely used of mathematical physiological and psychological models will be explained and reviewed.

Fanger comfort model: Overall Psychological Model

One for the most widely used thermal comfort models, the predicted mean vote (PMV) and predicted percentage dissatisfied (PPD) by Fanger, was designed for use in a uniformed environment(H. Zhang, Arens, Huizenga, & Han, 2010). That means environments with little temperature stratifications or temperature differences at the different parts of the body. However in many PV systems, especially those which target specified areas of the body, the PMV and PPD methods would be out of the scope it is designed for. Despite this, it was

shown in some studies, PMV is able to predict the overall sensation quite well (Park, Hellwig, Grün, & Holm, 2011).

The issue with the method proposed by Fanger is that the PPD is often not very accurate. This is similar to the perceived comfort level of the person. Studies have shown that in many cases, the comfort level of a certain environment has been perceived to be underestimated (van Hoof, 2008). For PV ventilation, PPD might not be sufficiently accurate to predict the comfort level.

PPD is developed based on the empirical data which Fanger obtained from experiments. It uses the PMV value calculated to determine the number of people who would be unsatisfied with the ambient conditions.

PMV and PPD main advantage is that it is a very simple way for engineers to have a rough estimate of the comfort level of the environment. PMV does not need finite element models or time differencing to obtain a prediction. However, it might not be as accurate as other type of models which considers the local sensations to calculate the overall sensation and thus the comfort level.

In fact what is interesting is how computer programs, especially thermal comfort software like RadTherm and Theseus FE, calculate the PMV. The surface temperature of the clothing is iterated using the equation developed by Fanger and the convection of the surface is calculated using a relationship between the clothing surface temperature and the ambient temperature of the air or the air velocity (International Standard, 2005). These 2 equations will be iterated until a convergence is reached. This means that any local values of the surface temperature and convection coefficients are not used. In fact some of the variables like velocity uses a area weighted average air velocity.

Fiala Models- Physiological model

Fiala models can be considered one of the most advanced physiological models. It consists of 2 parts, the passive and active systems. The passive system is similar to most physiological models catering to the thermal heat transfer in the body, through the tissues and the different models of heat transfer to the environment.

Figure 2-21: Passive and active systems in a complete thermoregulation model, extracted from (Fiala, Lomas, & Stohrer, 2001)

The active system will regulate the body responses to the ambient environment like the temperature, radiation, humidity and so forth. The body then responds with sweating, shivering, vasoconstriction or vasodilation based on certain temperature signals. This active system will serve as a feedback loop for the thermal regulation (Fiala et al., 2001).

The active system is designed based on experiments to understand the types of signals to track in order to trigger a certain response. For example to control shivering, the skin temperature and the head core temperature are tracked, depending on the difference between the skin temperature and the reference temperature, a certain amount of heat will be generated by shivering. This allows the model to behave accordingly to the temperatures experienced by the body.

As it is based on experimental data, the Fiala model can be considered a statistical mathematical model (Fiala et al., 2001). It is able to cater to common reactions to thermal experiences. Currently, major commercial thermal calculation packages like Radtherm and Theseus Fe use Fiala Models for human physiological calculations due to its comprehensive human model and the ability to cater to many situations such as transient and dynamic conditions.

Berkeley Models- Local and overall Psychological and Physiological models

University of California, Berkeley developed both psychological and physiological models. The physiological models are based on a model by Stolwijk which was developed in 1971. The Stolwijk model caters for 16 body parts and a total of 25 nodes (Romana et al., 2008). Each node is a data point in which a certain set of parameters can be found, such as temperature or heat transfer.

The Berkeley physiological model is an improvement of the Stolwijk model, adding features like unlimited segments and thus nodes which can be tracked, improved blood flow models and a clothing node catering to clothing worn by the person and allowing for conduction for surface contact. All effects are catered for scientifically (Huizenga, Hui, & Arens, 2001). This model also has an active and passive system.

The active system uses a certain set of equations with adjustable parameters to control the responses. These set of equations is dependable on the PMV and the clothing insulation of the body. When the PMV is of a certain value, a set of values of the parameters will be used to generate a response ie, shivering or sweating. The control for this system is considered simpler than the Fiala model.

The psychological model is based on experiments conducted with human subjects. The data is then collated and an empirical relationship is formed. This model is the result of numerous experiments on localised cooling on a number of body parts. Therefore, it would be suitable for PV simulations. The newer overall comfort model is one very accurate compared to the older model, and it is able to give very good overall comfort results. However, the mathematical formulation of the newer method is fairly complex, and difficult to implement (Foda et al., 2011). This is compared to the older method which uses a weighted method by

weighing the local sensation to obtain the overall sensation and then the comfort. Commercial software usually uses the older version of the Berkeley psychological model.

2-node pierce model- Physiological model

2-node pierce is a unique model used to calculate the temperatures in the body. This model is quite unlike the Berkeley and Fiala models and can be considered to be the simplest model of the three. Although it is also based on the Stolwijk model, the 2-node model only takes into consideration 2 nodes for each body segment cross section namely the skin and the core (Foda & Sirén, 2011). This is unlike more advanced models like Fiala which caters to even the bone muscle tissues and so forth.

The 2 node pierce model was designed by using temperature points of the skin in neutral condition. The core temperature would then be obtained using a line search method. This means that the core temperature is a control parameter then a physical property unlike the other models. This core temperature would then be a fixed or set parameter during the simulation of environmental conditions to calculate the skin temperatures and to control the body heat cooling or heat generation processes.

This model has shown to be one of the most accurate steady state models in obtaining the skin temperatures when a study on all the common physiological models was compared (Foda et al., 2011). Although more complex models like Fiala would be better at getting a better understanding of how the body will function and the temperature distribution through the tissue layers, the 2-node pierce model would be a simple model that allows for the study of comfort parameters. Its ability to capture dynamic behaviour is also good and comparable to the Fiala model.

Nilsson Equivalent Temperature- Local and Overall Psychological Model

Nilsson uses equivalent temperature to gauge the comfort of the person (Nilsson, 2007). Equivalent temperature is basically the temperature of air when all the humidity has been removed by an adiabatic process. It is said that this form of measure is closer to the temperatures that humans perceive compared to ordinary operative temperature with the influence of the non-evaporative heat loss.

From numerous test and experiments, Nilsson calculated the equivalent temperature for each comfortable level. For this scale, the skin temperatures of the person will be converted into the equivalent temperature and compared with the comfort and sensation scale to determine the comfort level for local parts or the overall body perception.

3 Simulation Governing Equations

3.1 Theory for heat transfer

Heat is transferred via 3 ways, namely conduction, convection and radiation. Some of the heat sources in a vehicle would be solar radiation, occupants in the car, auxiliary systems and so forth. The mathematical formulation of the 3 modes of heat transfer will be covered

In a tropical country like Singapore, one major portion of the heat comes from the sun. Rest will come from the occupants in the vehicle, followed by other external sources of heat like the ambient environment, the reflection of the ground and also heat given off from the absorption of the heat from the ground. Therefore, it would be important to ensure that these heat sources are studied and the thermal management system of the vehicle, including materials, design, air distribution and air conditioning system is suitable to cope with such heat loads.

Below is a diagram of how external heat can enter the cabin space. Solar radiation will come in 2 forms, one is direct from the direction of the sun, the other is diffused which can be considered to be coming from vertically downwards. The diffused form of solar radiation would be due to the scattering of the solar radiation as it hits the clouds or the atmosphere. However, in most case and especially in Singapore's case, the diffused radiation is only a small fraction of the total radiation which hits ground. Heat exchange with the environment can come in the form of radiation and convection depending on the interior surface temperature.

Solar radiation, a form of short range wave, which hits the windows can be absorbed, reflected or transmitted. In fact a vehicle works a lot like a greenhouse. Solar radiation will be absorbed by the window, heating up the windows. However, a major portion will be transmitted into the space. The materials in the cabin will absorb the solar radiation resulting in the rise of temperature of these surfaces. Heated surfaces will emit radiation via longwave radiation such as IR. Windows tend to have low transmissivity for longwave radiation resulting in these heat being trapped in the space unable to radiate out. Moreover the hot surfaces will heat up the cooler air via convection. This results in an increase in the cabin temperature unless this hot air is removed and cooler air is introduced.

Figure 3-1: Environmental heat loads on vehicle (extracted from Theseus FE manual)

Conduction

The basic equation for conduction in 3- dimensions is as follows:

$$\vec{q} = -\vec{k} \cdot \nabla T \quad (3-1)$$

The conservation of energy for thermal effects, for transient behaviour is:

$$\rho c \frac{\partial T}{\partial t} = \nabla(\vec{k} \nabla T) + \dot{Q} \quad (3-2)$$

Conduction into the material from the surface is the sum of the heat inputs in the surface. This includes convection, radiation or heat from a defined heat source on the surface.

$$q_{cond} = -\sum q_{bc} = -(q_{conv} + q_{rad} + q_{appl}) \quad (3-3)$$

Convection

The energy via convection on a surface is given by:

$$Q_{conv} = Ah_c(T_{surf} - T_f) \quad (3-4)$$

Forced convection heat transfer coefficient will be a function of fluid velocity. Likewise, for natural convection, the heat transfer coefficient will depend on the temperature difference between the surface and the fluid.

$$h_{c,for} = f(v_f)$$

$$h_{nat} = f(T_{surf} - T_f) \quad (3-5)$$

For mixed convection phenomena, one can either take the convection coefficient with the highest value, in order words dominant convection type. This means if force convection gives a higher value, then the convection phenomena is regarded at forced convection. If not the

mixed convection heat transfer coefficient can be a cube root of the sum of the cube of the forced and natural heat transfer coefficient.

$$h_c = h_{c,mix} = \sqrt[3]{h_{for}^3 + h_{nat}^3} \quad (3-6)$$

In order to calculate the heat transfer coefficient of a case, generally, the Nusselt number is found first. The Nusselt number is basically the ratio between the convection to the conductive heat transfer across the boundary. This means it give an understanding of the strength of convective heat transfer compared to the conductive heat transfer from the fluid to the surface.

$$Nu_L = \frac{h_L x}{k_f} \quad (3-7)$$

Prandtl number is one of the parameters used to determine the nusselt number. Prandtl number is defined as the ratio between the momentum diffusivity to the thermal diffusivity.

$$Pr = \frac{v c_p \rho}{k_f} \quad (3-8)$$

Reynold number basically expresses the ratio of the inertial forces to the viscous forces. It is used as a gauge to understand the flow behaviour, to determine the flow regime. When the Reynold number is above a critical value, it is assumed that the flow would be considered turbulent. Likewise if lower than a certain value, lamina flow is assumed. The critical value will depend on the situation, if it is a flat plate or a tube for example.

$$Re_x = \frac{v_f x}{\nu} \quad (3-9)$$

The local Nusselt numbers at a certain position on the flat plate or surface can be found using the following relations shown below:

$$\begin{aligned} Nu_{x,lam} &= 0.332 Re_L^{1/2} Pr^{1/3} & \text{for } Re_x < Re_{crit} = 5E5 \\ Nu_{x,turb} &= 0.0296 Re_x^{4/5} Pr^{1/3} & \text{for } Re_x > Re_{crit} = 5E5 \end{aligned} \quad (3-10)$$

For the Nusselt number over the whole plate, the local Nusselt can be integrated over the length of the plate or element. Depending on the behaviour of the flow, different local Nusselt numbers will be used.

$$\begin{aligned} Nu_{L,lam} &= 0.664 Re_L^{1/2} Pr^{1/3} & \text{for } Re_{x=0..L} < Re_{crit} \\ Nu_L &= (0.037 Re_L^{4/5} - 871) Pr^{1/3} & \text{for } Re_x > Re_{crit} \\ Nu_{L,turb} &= 0.037 Re_L^{4/5} Pr^{1/3} & \text{for } Re_{x=0..L} > Re_{crit} \end{aligned} \quad (3-11)$$

The Grashof number, Gr is the ratio of the buoyancy of fluid to the viscous forces acting on the fluid. The Rayleigh number essentially gives a value to be compared to the fluid's critical value. If the value exceeds the critical value, heat transfer can be stated to be convection. If the Rayleigh number is below the critical value, heat transfer mainly occurs by conduction through the fluid.

$$Ra_L = Gr_L Pr \quad \text{where } Gr_L = \frac{gL^3\beta(T_{surf} - T_f)}{\nu^2}$$

$$\beta = \frac{1}{273.15 + T_f} \quad (3-12)$$

As for natural convection, the orientation of the flat plate will also determine the behaviour of the flow. A vertical flat plate with an inclination less than 60° , the Nusselt number for natural convection is:

$$\text{Case A} \quad Nu_L = (0.825 + 0.3245Ra_L^{1/6})^2 \quad \text{where } Ra_L = Gr_L Pr_L \cos\theta \quad (3-13)$$

A horizontal plate with a heated top or cooled bottom:

$$\text{Case B} \quad Nu_{L,lam} = 0.54Ra_L^{1/4} \quad \text{for } 10^4 \leq Ra_L \leq Ra_{crit}$$

$$= 4.7384 \cdot 10^6$$

$$Nu_{L,lam} = 0.15Ra_L^{1/3} \quad \text{for } Ra_{crit} = 4.7384 \cdot 10^6 \leq Ra_L \leq 10^{11}$$

$$\leq 10^{11} \quad (3-14)$$

Similarly, for a horizontal plate with a heated bottom and a cooled top:

$$\text{Case C} \quad Nu_{L,lam} = 0.27Ra_L^{1/4} \quad \text{for } 10^5 \leq Ra_L \leq 10^{10} \quad (3-15)$$

For the horizontal plate cases, the characteristic length L is:

$$L = \frac{\text{area}}{\text{perimeter}} \quad (3-16)$$

There would be cases where the angle of the plate lies greater than 60° but it is not considered horizontal. In such cases, equations (3-17) can be used depending on whether it is case B or C:

$$h_{nat} = \sqrt{(h_a \cos\theta)^2 + (h_c \text{ or } h_b \sin\theta)^2}$$

$$h_{nat} = \sqrt{(h_a \cos\theta)^2 + (h_c \text{ or } h_b \sin\theta)^2} \quad (3-17)$$

Radiation

There are 2 types of wavelengths covered in the radiation portion, namely the long wavelength type and the short wavelength type. The long wavelength type will be used to cater to the thermal radiation spectrum while the short wavelength type will cover the solar radiation spectrum.

For the long wavelength properties of materials, it is generally considered that absorptivity of the material is equal to the emissivity of the material. This is known as Kirchoff's Law.

$$\alpha = \varepsilon \quad (3-18)$$

As for long wavelength, transmissivity is usually not considered as optically transparent such as windows are usually not transparent towards long wavelength radiation. Therefore, reflectivity and emissivity are the only 2 optical properties taken into account.

$$\rho = 1 - \varepsilon \quad (3-19)$$

For short range radiation, transmissivity is taken into account giving the following relationship:

$$\alpha + \rho + \tau = 1 \quad (3-20)$$

It is known that the optical or radiation properties of a material can differ at different angles of incidence. Therefore, the absorptivity of a material is shown as follows:

$$\begin{aligned} \alpha(\varphi) &= \alpha_0 & \text{for } \varphi < 60 \\ \alpha(\varphi) &= \alpha_0 \cdot 4 \cos(\varphi) [1 - \cos(\varphi)] & \text{for } \varphi > 60 \end{aligned} \quad (3-21)$$

In reality, the absorptivity, reflectivity and transmissivity of a material, especially materials that allow light through, will vary depending on the refractive index, n and the angle of incidence, φ . There is a mathematical relationship to be taken into consideration, n , φ and an additional variable known as the transmission index, t . This transmission index of the material can be obtained by measurements of the behaviour at zero degrees incidence and calculated.

The heat from the sun is dependent on the direct radiation from the sun, the diffused radiation from the sky and the reflectivity of the diffused radiation from the ground. Note that I_{gnd} consists of 2 types of radiation from the sun with relation to the ground, namely, the direct radiation portion and the diffused radiation portion.

$$q_{sun} = I_{direct} \cos(\varphi) + I_{df} F_{sky} + \rho_{gnd} I_{gnd} F_{gnd} \quad (3-22)$$

For the diffused radiation from the sky and the ground, the view factors can be easily found by using the view factors relationship of having an infinity large plane in relation to a plane at a certain angle β .

$$F_{sky} = \frac{[1 + \cos(\beta)]}{2} \quad F_{gnd} = \frac{[1 - \cos(\beta)]}{2} \quad (3-23)$$

The sun total energy hitting the surface is the sum of the absorptivity, reflectivity and transmissivity.

$$q_{sun,bc} = q_{sun,ab} + q_{sun,rf} + q_{sun,tr} \quad (3-24)$$

Each type of heat energy input is dependent on the direct radiation and the diffused radiation from the sky and the ground.

$$\begin{aligned} q_{sun,ab} &= \alpha(\varphi) I_{dr} \cos(\varphi) + \alpha(\varphi_{eff,sky}) I_{df} F_{sky} + \alpha(\varphi_{eff,gnd}) I_{gnd} F_{gnd} \\ q_{sun,tr} &= \tau(\varphi) I_{dr} \cos(\varphi) + \tau(\varphi_{eff,sky}) I_{df} F_{sky} + \tau(\varphi_{eff,gnd}) I_{gnd} F_{gnd} \\ q_{sun,ab} &= \rho(\varphi) I_{dr} \cos(\varphi) + \rho(\varphi_{eff,sky}) I_{df} F_{sky} + \rho(\varphi_{eff,gnd}) I_{gnd} F_{gnd} \end{aligned} \quad (3-25)$$

The above equations are actually targeted at external surface radiation. To study the sun's exposure to internal elements, the transmitted solar radiation and reflected solar radiation from all other cells have to be considered and summed up. Superscript k is the element to be

calculated and n is the surface transmitting radiation through. Radiation from the other cells which transmits diffused radiation and or reflects radiation will be summed up.

$$q_{sun,bc}^k = \tau(\varphi^n) I_{dr} \cos(\varphi^k) + \sum F Q_{sun,tr,df} + \sum F Q_{sun,rf} \quad (3-26)$$

The standard heat transfer via long wave radiation is given below:

$$Q_{1-2} = \varepsilon_1 \varepsilon_2 \sigma (T_1^4 - T_2^4) F_{12} dA_1 \quad (3-27)$$

The view factor for internal cells is quite different. The basic equation is shown below, however, there are various ways to solve this equation using different numerical methods such as the surface to surface integration method or the Hemicube method.

$$F_{12} = \frac{1}{\pi A_1} \int_{A_1} \int_{A_2} \frac{\cos \varphi_1 \cdot \cos \varphi_2}{(s_{12})^2} dA_2 dA_1 \quad (3-28)$$

3.2 Basic Governing Equations for CFD

For StarCCM, like most other CFD packages or solvers, there are 3 main governing equations including a series of supporting equations. The equations are as follows:

Continuity

$$\frac{\partial \rho}{\partial t} + \frac{\partial \rho u_i}{\partial x_i} = 0 \quad (3-29)$$

However, as for the simulations, it is assumed that the air is incompressible:

- ρ is a constant

This results in equation (3-30) becoming:

$$\frac{\partial u_i}{\partial x_i} = 0 \quad (3-30)$$

As Reynolds-averaging is done, the following relationship is used:

$$u_i = U_i + u'_i \quad (3-31)$$

Therefore:

$$\frac{\partial U_i}{\partial x_i} = 0 \quad (3-32)$$

Momentum (Navier Stokes)

$$\frac{\partial \rho u_i}{\partial t} + \frac{\partial \rho u_i u_j}{\partial x_j} = -\frac{\partial P}{\partial x_i} + \frac{\partial}{\partial x_j} \left[\mu \left(\frac{\partial u_i}{\partial x_j} \right) \right] + \rho g_i + S_{M_i} \quad (3-33)$$

However as for the simulations, it is assumed that the air is incompressible:

- Mass continuity, $\text{div } \mathbf{u} = 0$,
- Second viscosity effect, $\lambda = 0$,
- Viscosity μ is a constant, and
- Density ρ is a constant.

results in equation (3-33) becoming:

$$\rho \frac{\partial u_i}{\partial t} + \rho \frac{\partial u_i u_j}{\partial x_j} = -\frac{\partial P}{\partial x_i} + \mu \frac{\partial}{\partial x_j} \left(\frac{\partial u_i}{\partial x_j} \right) + \rho g_i + S_{M_i} \quad (3-34)$$

As Reynolds-averaging is done, the same relationship (3-31) is used. Therefore:

$$\rho \frac{\partial U_i}{\partial t} + \rho \frac{\partial U_i U_j}{\partial x_j} + \rho \frac{\partial \overline{u'_i u'_j}}{\partial x_j} = -\frac{\partial P}{\partial x_i} + \mu \frac{\partial}{\partial x_j} \left(\frac{\partial U_i}{\partial x_j} \right) + \rho g_i + S_{M_i} \quad (3-35)$$

Energy

$$\frac{\partial \rho c_p T}{\partial t} + \frac{\partial \rho c_p T u_j}{\partial x_j} = \frac{\partial}{\partial x_j} \left(k \frac{\partial T}{\partial x_j} \right) + S_E \quad (3-36)$$

However, as for the simulations, it is assumed that the air is incompressible:

- ρ is a constant
- c_p is a constant

This results in equation (3-36) becoming:

$$\rho c_p \frac{\partial T}{\partial t} + \rho c_p \frac{\partial T u_j}{\partial x_j} = \frac{\partial}{\partial x_j} \left(k \frac{\partial T}{\partial x_j} \right) + S_E \quad (3-37)$$

As Reynolds-averaging is done, the same relationship (3-31) is used. Therefore:

$$\rho c_p \frac{\partial T}{\partial t} + \rho c_p \frac{\partial T u_j}{\partial x_j} = \frac{\partial}{\partial x_j} \left(k \frac{\partial T}{\partial x_j} \right) + \rho c_p \left[-\frac{\partial \overline{u'_i T'}}{\partial x_i} \right] + S_E \quad (3-38)$$

where ρ is the density, μ is the dynamic viscosity, u is the velocity, P is the pressure, T is the temperature, t is the time, x_i is the x , y or z position tensor, u_i is the x , y or z velocity tensor, c_p is the specific heat capacity and k is the thermal conductivity.

Incompressible Fluid with Buoyancy

Buoyancy driven flows are due to differing densities within the fluid. It is dependent on the body forces such as gravity. The fluid becomes less dense due to the fluid particles becoming closer to each other. This is usually due to a change in temperature and thus a change in internal energy of the particle. The changes in density subjected to external forces would lead to the fluid flow, i.e. denser fluid moving in the direction of gravity. Likewise if the fluid is less dense, it will move against the direction of gravity. A fluid flow affected dominantly by temperature in this manner is known as free convection. Although, the air in the computational domain is considered to be incompressible, the density of the fluid is still affected by the temperature. Density is governed by the following equation:

$$b = g(\rho_0 - \rho)/\rho_0 \quad (3-39)$$

The reference pressure in hydrostatic balance with the reference density

$$P_0 = -g\rho_0 z + \text{constant} \quad (3-40)$$

Next, substitute P with $\tilde{P}$ with the following relation:

$$\tilde{P} = P - P_0(z) \quad (3-41)$$

With reference to momentum equation (3-33), note that as gravity usually only acts on the z axis:

$$-\frac{\partial P}{\partial x_i} + \rho g_i = \begin{pmatrix} -\frac{\partial P}{\partial x} \\ -\frac{\partial P}{\partial y} \\ -\frac{\partial P}{\partial y} - g\rho \end{pmatrix} = \begin{pmatrix} -\frac{\partial \tilde{P}}{\partial x} \\ -\frac{\partial \tilde{P}}{\partial y} \\ -\frac{\partial \tilde{P}}{\partial y} + g\rho_0 - g\rho \end{pmatrix} \quad (3-42)$$

With the following relationship for

Note that:

$$\alpha = \frac{1}{V} \frac{\partial V}{\partial T} = -\frac{1}{\rho} \frac{\partial \rho}{\partial T} \approx -\frac{1}{\rho} \frac{\rho_0 - \rho}{T_0 - T} \quad (3-43)$$

This allows for thermal expansion coefficient to be described by temperature.

This results in the following relationship:

$$\rho_0 - \rho \approx \rho \alpha (T - T_0) \quad (3-44)$$

Substituting equation (3-44) into (3-42):

$$\begin{pmatrix} -\frac{\partial \tilde{P}}{\partial x} \\ -\frac{\partial \tilde{P}}{\partial y} \\ -\frac{\partial \tilde{P}}{\partial y} + g\rho_0 - g\rho \end{pmatrix} = \begin{pmatrix} -\frac{\partial \tilde{P}}{\partial x} \\ -\frac{\partial \tilde{P}}{\partial y} \\ -\frac{\partial \tilde{P}}{\partial y} + \rho \alpha (T - T_0) \end{pmatrix} = -\frac{\partial \tilde{P}}{\partial x_i} + \rho b_z \quad (3-45)$$

This results in the following momentum equation with the buoyancy term with respect to temperature to be:

$$\rho \frac{\partial u_i}{\partial t} + \rho \frac{\partial u_i u_j}{\partial x_j} = -\frac{\partial \tilde{P}}{\partial x_i} + \mu \frac{\partial}{\partial x_j} \left(\frac{\partial u_i}{\partial x_j} \right) + \rho b_z + S_{M_i} \quad (3-46)$$

3.2.1 Turbulence Models

For these series of simulations, the *Realisable* $K - \varepsilon$ is used. This is due to its widespread use for indoor simulations. In fact the $K - \varepsilon$ turbulence models are the mostly used for indoor airflow simulations such for vehicle cabins and office space (Zhai, Zhang, Zhang, & Chen, 2007).

These models are essential to give a relatively accurate behaviour of the fluid flow in the enclosed space. A well ventilated space would have a positive effect on the well-being and health of the people working or staying in this space. $K - \varepsilon$ turbulence models have been shown to be able to simulate quite accurately, the behaviour of the fluid ie, air allowing for studies on the particulate movement, humidity control, concentration of contaminants, air velocity and temperature, and turbulence intensity. It is suitable mainly for non-boundary governed flows and thus suitable.

$K - \varepsilon$ turbulence models consist mainly of 2 transport equations, one for the turbulent dissipation, k and the other for the turbulent kinetic energy, ε .

With reference to equation (3-35),

$$\rho \frac{\partial U_i}{\partial t} + \rho \frac{\partial U_i U_j}{\partial x_j} + \rho \frac{\partial \overline{u'_i u'_j}}{\partial x_j} = -\frac{\partial P}{\partial x_i} + \mu \frac{\partial}{\partial x_j} \left(\frac{\partial U_i}{\partial x_j} \right) + \rho g_i + S_{M_i} \quad (3-35)$$

Note that term 3, $\overline{u'_i u'_j}$ is known as the Reynold's stress tensor. It describes the correlation between the fluctuating velocities of the turbulent flow. The relationship between the stress tensor and the turbulent flow needs to be found in order to solve the system of equations. The stress tensor can be described with the use of the Boussinesq Hypothesis.

Boussinesq Hypothesis

The Boussinesq Hypothesis states the following, with reference to the term 3 in equation (3-35):

$$-\rho \overline{u'_i u'_j} = \mu_t \left(\frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right) - \frac{2}{3} \rho \delta_{ij} k \quad (3-47)$$

where k is the turbulent energy and δ_{ij} is the Kronecker Delta.

k can be described as:

$$k = \frac{1}{2} \overline{u'_i u'_i} \quad (3-48)$$

The eddy viscosity, μ_t can be found in the following relationship:

$$\mu_t = C_\mu \rho q l_m \quad (3-49)$$

Using the analogy to solve the stress tensor for the momentum equation, the same idea can be extended to turbulent heat fluxes for equation (3-38). This would be the 2nd term on the right hand side:

$$\rho c_p \frac{\partial T}{\partial t} + \rho c_p \frac{\partial T U_j}{\partial x_j} = \frac{\partial}{\partial x_j} \left(k \frac{\partial T}{\partial x_j} \right) + \rho c_p \left[-\frac{\partial \overline{u'_i T'}}{\partial x_i} \right] + S_E \quad (3-50)$$

The 2nd term can be described with the following relation:

$$\overline{u'_i T'} = -\alpha_t \frac{\partial T}{\partial x_i} \quad (3-51)$$

where $\alpha_t = \mu_t / \sigma_t$ and σ_t is the turbulent Prandtl number. The Prandtl number is usually taken as a constant ranging from 0.7 to 0.9 depending on the type of flow (0.7 for free flow and 0.9 for near-wall).

K-epsilon Realizable

The simulations performed in this project use the $k - \varepsilon$ realizable turbulence model. In order to solve the eddy viscosity term μ_t mentioned above, the 3 relations are used.

With reference to (3-49):

$$\mu_t = C_\mu \rho q l_m = C_\mu \rho \frac{k^2}{\varepsilon} \quad (3-52)$$

The 2 transport equations for k and ε are:

$$\frac{\partial(\rho k)}{\partial t} + \frac{\partial(\rho k u_j)}{\partial x_j} = \frac{\partial}{\partial x_j} \left[\left(\mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right] + G_k + G_b - \rho \varepsilon + S_k \quad (3-53)$$

$$\begin{aligned} \frac{\partial(\rho \varepsilon)}{\partial t} + \frac{\partial(\rho \varepsilon u_j)}{\partial x_j} &= \frac{\partial}{\partial x_j} \left[\left(\mu + \frac{\mu_t}{\sigma_\varepsilon} \right) \frac{\partial \varepsilon}{\partial x_j} \right] + \rho C_1 S \varepsilon - \rho C_2 \frac{\varepsilon^2}{k + \sqrt{\nu \varepsilon}} \\ &\quad + C_{1\varepsilon} \frac{\varepsilon}{k} C_{3\varepsilon} G_b + S_\varepsilon \end{aligned} \quad (3-54)$$

where $C_1 = \max \left[0.43, \frac{\eta}{\eta + 5} \right]$, $\eta = S \frac{k}{\varepsilon}$, $S = \sqrt{2 S_{ij} S_{ij}}$, $S_{ij} = \frac{1}{2} \left(\frac{\partial u_j}{\partial x_i} + \frac{\partial u_i}{\partial x_j} \right)$

G_b is the generation of turbulence kinetic energy due to buoyancy. G_b is given by:

$$G_b = -g_i \cdot \frac{\mu_t}{\rho Pr_t} \frac{\partial \rho}{\partial x_i} \quad (3-55)$$

where Pr_t , the turbulent Prandtl number for energy (usually $Pr_t = 0.85$) and g_i is the component of the gravitation vector in the i direction.

G_k is the generation of turbulence kinetic energy due to buoyancy. G_k is given by, when formulated consistent with the Boussinesq hypothesis:

$$G_k = \mu_t S^2 \quad (3-56)$$

Wall functions

The wall functions are used to approximate the behaviour and properties of the fluid at the boundary layer near a wall. The advantage of such a function is that the mesh at this region would not need to be so fine, effectively decreasing the computation time required. Wall functions effective, when the boundary layer is not of utmost importance. The wall function relations shown here are the general ones, further details into the functions describing the turbulence and lamina effects at the wall will not be covered here.

The wall functions will simulate the fluid flow in accordance to the logarithmic region of the velocity profile at the boundary layer. The relationship is given by:

$$u^+ = \frac{\bar{v}_t}{u_\tau} = \frac{1}{\kappa} \ln(y^+) + B \quad (3-57)$$

where $\bar{v}_t$ is the mean velocity parallel to the wall. u_τ is the shear velocity, given by $u_\tau = \sqrt{|\tau_w|/\rho}$. κ is the karman constant equals to 0.41. B is an empirically derived constant related to the thickness of the viscous sub layer. y^+ is the dimensionless distance from the wall.

y^+ is given by:

$$y^+ = \frac{\rho u_\tau y}{\mu} \quad (3-58)$$

As for the turbulence at the wall, the assumption for the turbulent kinetic energy is that the production of turbulence and dissipation is equal. This results in:

$$k = \frac{u_\tau^2}{C_\mu} \quad (3-59)$$

3.3 Theory of on Virtual Human Manikin

The simulation couples the use of the virtual human manikin to understand the comfort of the human manikin. This manikin was developed by Fiala (Dusan, Lomas, & Stohrer, 1999) for the study of personal ventilation. American Society of Heating, Refrigeration, and Air-Conditioning Engineers (ASHRAE) and International Organization for Standardization (ISO) have a widely used comfort chart that determines the comfort level of a room depending on the humidity and temperature, however, this chart is mainly for a room with relatively uniformed temperatures.

Zhang has shown that with selected cooling with a warmer ambient surrounds at the certain part of the human body, it may bring about a comfort level similar to the one with a uniformed cool surroundings. In fact this is particularly true for a vehicle where the occupants are exposed to localised heating and cooling depending on factors like sun position and how far they are from the aircon for example.

Zhang's manikin is based on her studies done with a group of people of varying genders and ages. Each person was exposed to cooling at various parts of the body, and they were asked on the sensation and the overall comfort of the cooling at that part. All the empirical data were compiled and used in the design of the virtual manikin. This is one of the most complete virtual manikin created at current state of the art.

The Virtual human Manikin is made up 2 major parts, the passive system and the active system.

3.3.1 Passive System

The passive system is the basic model of the manikin. The passive system models interaction of the body with the environment via heat transfer effects such as conduction, convection and radiation. Therefore based on the environment temperature, vapour pressure and heat loads, the body will respond likewise to achieve an energy balance to maintain its core temperature.

The manikin used in the simulation is one that has 15 body elements. The model discretized into elements with 3 points per material layer i.e. skin muscle. Therefore each material layer will have 2 sub-layers. The manikin is made to mimic an actual human. As such it consists of a core with bone material followed by muscle layer, than fat and lastly the skin layer. An additional cloth layer is used to mimic clothing.

Note that shell model with many surface elements, the mesh model seen in Theseus is only for pre and post processing, and radiation calculations. It is the inner manikin consisting of only 15 elements which will calculate the body responses to heat.

3.3.1.1 Heat transfer in tissue

Figure 3-2: Cross section of a left leg with the nodes (extracted from Dusan et al., 1999)

The heat equation to describe the heat transfer rate in the living tissues:

$$k \left(\frac{\partial^2 T_i}{\partial r^2} + \frac{\omega}{r} \frac{\partial T_i}{\partial r} \right) + q_m + \rho_{bl} w_{bl} c_{bl} (T_{bl,a} - T_i) = \rho c \frac{\partial T_i}{\partial t} \quad (3-60)$$

where:

ρ	: Density
c	: Specific heat
T	: Temperature
t	: Time
k	: Conductivity
r	: Body radius
q_m	: Metabolic heat
ρ_{bl}	: Density of blood
w_{bl}	: Blood perfusion rate (<i>volume/kg.s</i>)
c_{bl}	: Specific heat of blood
$T_{bl,a}$	: Temperature of arterial blood
ω	: 1 for cylindrical body element, 2 for spherical body element

This is the governing equation which solves for the temperatures in tissue layers or nodes. The other relations would be used to find the terms in this temperature equation.

3.3.1.2 Metabolic Heat

q_m is calculated using:

$$q_m = q_{m,bas,0} + \Delta q_m \quad (3-61)$$

where:

- $q_{m,bas,0}$: Basal value of heat production based on environment
- Δq_m : Heat production due to exercising or shivering

For muscle layers, the additional heat production contains three components:

$$\Delta q_m = \Delta q_{m,bas} + q_{m,sh} + q_{m,w} \quad (3-62)$$

The tissue setpoint temperature is T_0 . However due to local regulations, it results in temperature variations of the tissue temperature T . This results in a heat transfer with heat production.

$$\Delta q_{m,bas} = \Delta q_{m,bas,0} + [2^{(t-t_0)/10} - 1] \quad (3-63)$$

$q_{m,sh}$ from equation (3-62) one part of the regulatory response function, Sh controlled by the active system.

$$\Delta q_{m,sh} = a_{sh,j} \frac{Sh}{V_{mus,j}} \quad (3-64)$$

where

- $a_{sh,j}$: Shivering distribution coefficient for body element j
- $V_{mus,j}$: Muscle volume of element j
- Sh : Regulatory response function Sh ($Sh = 0$ when pure passive system considered)

Metabolism due to exercise takes a similar form to shivering

$$\Delta q_{m,w} = a_{w,j} \frac{H}{V_{mus,j}} \quad (3-65)$$

where the workload H is given by:

$$H = act \frac{M_{bas,0}}{act_{bas}} (1 - \eta) - M_{bas,0} \quad (3-66)$$

where:

- act_{bas} : act_{bas} is the measure of the activity level at basal value, equal to 0.8 met at thermal neutrality
- $M_{bas,0}$: $M_{bas,0}$ is taken as a constant at 87.1 W
- act : act is taken as 1 met for a car driving person
- $1 - \eta$: Represent the fraction of the global metabolic heat production that results in a human heating

The η is the mechanical efficiency of a human:

$$\eta = \max[0, 0.2 \cdot \tanh(0.39 \cdot act - 0.6)] \quad (3-67)$$

3.3.1.3 Blood circulation

Blood circulation is an import aspect of the simulation as it serves as a source or sink for heat transfer. Certain parts of the body might reach unrealistic temperatures without the blood circulation simulated.

Heat exchanged between the adjacent arteries and veins is given by:

$$Q_x = h_x (T_{bl,a} - T_{bl,v}) \quad (3-68)$$

where

- h_x : Individual heat transfer coefficient
- $T_{bl,a}$: Temperature of blood of arteries
- $T_{bl,v}$: Temperature of blood of veins

$$T_{bl,v} = \frac{\int w_{bl} T dV}{\int w_{bl} dV} \quad (3-69)$$

And arterial temperature:

$$T_{bl,a} = \frac{\rho_{bl} c_{bl} \int w_{bl} dV \cdot T_{bl,p} + h_x \cdot T_{bl,v}}{\rho_{bl} c_{bl} \int w_{bl} dV + h_x} \quad (3-70)$$

The blood pool temperature $T_{bl,p}$ is represented with additional degree of freedom and derived from the following equation:

$$T_{bl,p} = \frac{\sum_j \left[\frac{\rho_{bl} c_{bl} \int w_{bl} dV_j}{\rho_{bl} c_{bl} \int w_{bl} dV_j + h_{x,j}} \cdot \rho_{bl} c_{bl} \int T \cdot w_{bl} dV_j \right]}{\sum_j \left[\frac{(\rho_{bl} c_{bl} \int w_{bl} dV_j)^2}{\rho_{bl} c_{bl} \int w_{bl} dV_j + h_{x,j}} \right]} \quad (3-71)$$

where $\sum_i$ will run for all the body elements

$$w_{bl} = w_{bl,0} + \Delta w_{bl,0} \quad \text{or} \quad \beta = \beta_0 + \Delta\beta \quad (3-72)$$

where $\beta = \rho_{bl} c_{bl} w_{bl}$

For non-skin layers, extra metabolism occurs due to an increase need of oxygen.

$$\Delta\beta = \mu_{bl} \cdot \Delta q_m \quad \text{where } \mu_{bl} \text{ is } 0.932 \quad (3-73)$$

With the influence of the active systems, functions Cs and DI are considered for skin blood flow

$$\beta' = \int \beta dV_{sk} = \frac{\beta'_0 + a_{di,j} \cdot DI}{1 + a_{cs,j} \cdot Cs \cdot e^{-DI/80}} \cdot 2^{(T-T_0)/10} \quad (3-74)$$

Which is had strong dependency on environmental conditions, where:

- $a_{di,j}$: Vasodilation distribution coefficient (passive system $DI = 0$)
- $a_{cs,j}$: Vasoconstriction distribution coefficient (passive system $Cs = 0$)
- T_0 : Setpoint for thermal neutrality

3.3.1.4 Heat Exchange with Environment

The mathematical model for heat exchange with the environment for the manikin is very similar to the one use for the other objects in the computational space. The heat transfer means are similar, namely, conduction via contact with the manikin i.e. car seat, convection via the air around the manikin, and radiation from various surfaces in the vehicle. However for the manikin one additional process for heat transfer would be via heat evaporation. Moreover as the manikin breathes, heat is lost by convection and evaporation.

This section will only cover the heat transfer that is unique to the manikin compared to other surfaces.

Wet Heat Loss Insulation

Wet heat loss is heat transfer through the clothes driven by evaporation between the skin and ambient environment:

$$q_{e,cl} = -U_{e,cl}(p_{sk} - p_a) \quad (3-75)$$

where

- p_{sk} : Water vapour pressure at skin surface
- p_a : Vapour pressure of ambient air
- $U_{e,cl}$: Overall evaporative heat transfer coefficient

The overall evaporative heat transfer coefficient is given by:

$$U_{e,cl} = \frac{1}{\sum_{lay} R_{e,cl,lay} + \frac{1}{f_{cl} \cdot h_{c,mix} \cdot L_a}} \quad (3-76)$$

where

- L_a : Lewis constant = 0.0165 K/Pa
- $R_{e,cl,lay}$: Local evaporative resistance for each clothing layer
- f_{cl} : Clothing surface area factor

The clothing surface area factor is given by:

$$f_{cl} = \frac{A_{sf}}{A_{sk}} \quad (3-77)$$

Evaporative

Evaporative heat loss is related to steam mass loss and given by:

$$\frac{q_e}{\lambda_{H_2O}} = \frac{\partial m}{\partial t} \cdot \frac{1}{A_{sk}} \quad (3-78)$$

where $\lambda_{H_2O} = 2256 \text{ kJ/kg}$ which is the latent heat of vaporisation

The rate of mass lost per sector skin area A_{sk} consists of two parts:

$$\frac{\partial m}{\partial t} \cdot \frac{1}{A_{sk}} = -\frac{\partial m_{sk}}{\partial t} \cdot \frac{1}{A_{sk}} - \frac{p_{sk,sat} - p_{sk}}{\lambda_{H_2O} R_{e,sk}} \quad (3-79)$$

where

$p_{sk,sat}$	:	The saturated vapour pressure at the outer skin layer of the body element
p_{sk}	:	Actual vapour pressure at skin surface
$1/R_{e,sk}$	:	Skin moisture permeability= $0.003 \text{ W/m}^3 \text{ Pa}$
$\frac{\partial m_{sk}}{\partial t}$	:	Rate of change of mass of water vapour lost from skin
$\frac{\partial m_{sk}}{\partial t} \cdot \frac{1}{A_{sk}}$	:	Rate of mass by sweating
$\frac{p_{sk,sat} - p_{sk}}{\lambda_{H_2O} R_{e,sk}}$	:	Rate of mass by diffusion

As equation (3-75) will give the $q_{e,cl}$ due to wet heat loss, the unknown p_{sk} needs to be found. This can be found by substituting $\frac{\partial m}{\partial t} \cdot \frac{1}{A_{sk}}$ in equation (3-79) with equation (3-78) and q_e with $q_{e,cl}$ of equation (3-75). This will allow p_{sk} to be found.

$$\frac{\partial m_{sw}}{\partial t} = a_{sw,j} \frac{A_{sk,i}}{A_{sk,j}} S_w \cdot 2^{(T-T_0)/10} \quad (3-80)$$

where

$a_{sw,j}$	:	Sweating distribution coefficient
$A_{sk,i}$	:	Skin area of sector i
$A_{sk,j}$	:	Skin area of the body element j
T_0	:	Setpoint temperatures at thermal neutrality
S_w	:	Active Sweating response coefficient ($S_w = 0$ if passive system only)

Moisture Storage

The rate of moisture storage on skin surface dm_{ac}/dt is equivalent to the rate of moisture production dm_{sw}/dt minus the rate of moisture evaporation, when the vapour pressure at the skin surface p_{sk} exceeds the saturated vapour pressure $p_{sk,sat}$. This would mean that the vapour at the skin surface will evaporate into the ambient air.

$$\frac{\partial m_{ac}}{\partial t} \cdot \frac{1}{A_{sk}} = \frac{\partial m_{sw}}{\partial t} \cdot \frac{1}{A_{sk}} - \frac{U_{e,cl}(p_{sk,sat} - p_a)}{\lambda_{H_2O}} \quad (3-81)$$

The maximum value for m_{ac}/A_{sk} is limited at 35 g/m^2 . Excess moisture will be taken as surface runoff. If the skin is wet ($m_{ac} > 0$), the simulation model is set at $p_{sk} = p_{sk,sat}$. If sweating is stopped:

$$\frac{\partial m_{ac}}{\partial t} \cdot \frac{1}{A_{sk}} = - \frac{U_{e,cl}(p_{sk,sat} - p_a)}{\lambda_{H_2O}} \quad (3-82)$$

Respiration

Respiration is another path for heat transfer. It consists of a convective and an evaporative portion.

The convective part is given by:

$$\dot{C}_{rsp} = -1.948 \cdot 10^{-3} \cdot Q_m \cdot (32.6 - 0.934 \cdot T_a + 1.96 \cdot 10^{-4} \cdot p_a) \quad (3-83)$$

The evaporative part is given by:

$$\dot{E}_{rsp} = -4.373 \cdot Q_m \cdot (0.0277 - 6.5 \cdot 10^{-5} \cdot T_a - 4.91 \cdot 10^{-6} \cdot p_a) \quad (3-84)$$

Thus, the heat production by respiratory is given by:

$$\dot{Q}_{rsp} = \dot{C}_{rsp} + \dot{E}_{rsp} \quad (3-85)$$

where:

- Q_m : Actual metabolic heat production
- T_a : Air temperature
- P_a : Partial vapour pressure of ambient air

$\dot{Q}_{rsp}$ is distributed over body elements, the following is followed: 25% outer face muscles, 20% inner face muscles, 25% muscle band of neck and 30% lungs.

Contact

Thermal contact would couple the manikin skin area to the contact surface such as a seat for the transfer of heat. When a region is assigned for contact, the surface will not be subjected to other heat transfer methods such as convection or radiation.

Boundary Conditions of the heat transfer equation

The heat transfer equation (3-60) will be used to calculate the temperature of the discretised nodes of each layer of tissue from the bone to the skin surface. The 2 closure equations which would form the boundary conditions, one will describe the thermal storage term in the bone, the other will describe the heat transfer to the clothes and the ambient air.

For the thermal storage term for the bone, there are no prior nodes as it would be the first node positioned at the centre of the body structure. As such it would serve as a thermal storage region; however the temperature would still be controlled by the available blood vessels in the region.

$$k \left(\frac{\partial^2 T_i}{\partial r^2} + \frac{\omega}{r} \frac{\partial T_i}{\partial r} \right) + q_m + \rho_{bl} w_{bl} c_{bl} (T_{bl,a} - T_i) = \frac{\kappa}{\pi r^1} \sum_{k=1}^{sectors} \varphi_k q_{ifc,k} \quad (3-86)$$

where

$$q_{ifc} = \frac{k_r (T_- - T_+)}{r_{ifc} \ln \left(\frac{r_{ifc} + \frac{\Delta r}{2}}{r_{ifc} - \frac{\Delta r}{2}} \right)} \quad (3-87)$$

At the interface between the bone and the muscle tissue, q_{ifc} will be calculated using T_- at the bone and T_+ at the first node in the muscle tissue. This will serve the boundary condition for the bone side of the heat transfer, the first node.

As for the skin surface which is where the final node is situated, similar to equation (3-86) is used, taking q_{sk} at the skin is used as the same form as q_{ifc} . However in order to have a complete closure of the system of equations, one more equation is need. This would be the boundary condition for the skin heat transfer at the skin.

$$q_{sk} = U_{cl} (T_{sk} - T_{com}) + U_{e,cl} (P_{sk} - P_{air}) \quad (3-88)$$

where T_{com} is the combined operative environmental temperature which is used to combine the influences of convection, radiation and short-wave radiation absorbed by the body surface.

Likewise, it is important to note that U_{cl} incorporates the 2 heat transport mechanisms for a body, conduction through the clothes, surface convection to the ambient air and long wave thermal radiation to surroundings.

$$T_{com} = \frac{h_{c,mix} T_{air} + h_R T_{sr,m} + F_{sf,sr} \alpha_{sf} S}{h_{c,mix} + h_R} \quad (3-89)$$

3.3.1.5 Thermal Neutrality

Thermal Neutrality is the state of optimum thermal comfort for a manikin with no clothes. This would mean no active system regulatory responses like shivering or sweating occurs to cool and stabilize the manikin. The thermal neutrality would give the setpoint temperatures T_0 . The setpoint temperatures are found by running the manikin model at 30°C ambient temperature at 0.1 m/s air velocity. This will give the temperature of the body parts at thermal neutrality.

With these temperatures, when the signal is on, the active thermoregulation system will be switched on to adjust the temperature of the body. As mentioned earlier, this will be thermo-responses like shivering, sweating etc.

3.3.2 Active systems

The body responses is shown by research (Dusan, Lomas, & Stohrer, 2001) to be based on 3 major components, the temperature of the hypothalamus, T_{hy} (or in other words the head

region), the mean skin temperature, $T_{sk,m}$ (average from all the body parts), and lastly the rate of change of skin temperature with time. The rate of change of skin temperature with time will mean that the previous experienced temperature will affect the body response to existing conditions.

Therefore depending on the difference between the temperature experienced at thermal neutrality and temperature at the moment, this will trigger and control the magnitude of the thermal responses through global response functions. The global response function is in this form:

$$F = f\left(\Delta T_{hy}, \Delta T_{sk,m}, \frac{\partial T_{sk,m}}{\partial t}\right) \quad (3-90)$$

where

$$T_{sk,m} = \sum_j \left[a_{sk,i} \sum_i \frac{A_i}{A_j} T_{sk,j} \right] \quad (3-91)$$

Note that if F greater than 0, that would mean the response is active. The greater the number, the stronger the response. If $F = 0$, this would mean that there is no response. F cannot be negative though.

3.3.2.1 Global Vasoconstriction Function

This response occurs when the environment is very cold with results in limited blood flow to the parts of the body.

$$Cs = 35 \cdot [\tanh(0.29 \cdot \Delta T_{sk,m} + 1.11) - 1] \cdot \Delta T_{sk,m} \quad (3-92)$$

$$-7.7 \cdot \Delta T_{hy} + 3 \cdot \Delta T_{sk,m} \underbrace{\frac{\partial T_{sk,m}}{\partial t}}_{\text{function} = 0 \text{ if } \frac{\partial T_{sk,m}}{\partial t} > 0 \text{ or } \Delta T_{sk,m} > 0}$$

3.3.2.2 Global Vasodilatation Function

This response unlike the vasoconstriction, will lead to the rise of local blood flow at the skin. This appears when the environment is warm.

$$DI = 16 \cdot [\tanh(1.92 \cdot \Delta T_{sk,m} - 2.53) + 1] \cdot \Delta T_{sk,m} \quad (3-93)$$

$$+ 30 \cdot [\tanh(3.51 \cdot \Delta T_{hy} - 1.48) + 1] \cdot \Delta T_{hy}$$

3.3.2.3 Global Sweating Function

Sweating will produce sweat liquid on the skin. This will lead to evaporative heat loss due to the evaporation of sweat liquid. The liquid is limited to only 30 g/min and usually only occurs when the environment is warm.

$$SW = 0.65 \cdot [\tanh(0.82 \cdot \Delta T_{sk,m} - 0.47) + 1.15] \cdot \Delta T_{sk,m} \quad (3-94)$$

$$+ [5.6 \cdot [\tanh(3.14 \cdot \Delta T_{hy} - 1.83) + 6.4] \cdot \Delta T_{hy}$$

3.3.2.4 Global Shiver Function

Shivering increases the metabolic heat and creates the heat production in the muscle region. However the metabolic heat from shivering is limited to 350W only.

$$Sh = 10 \cdot [\tanh(0.51 \cdot \Delta T_{sk,m} + 4.19) - 1] \cdot \Delta T_{sk,m} - 27.5 \Delta T_{hy} \quad (3-95)$$

$$-28.2 + 1.9 \cdot \Delta T_{sk,m} \frac{\partial T_{sk,m}}{\partial t}$$

function = 0 if $\frac{\partial T_{sk,m}}{\partial t} > 0$ or $\Delta T_{sk,m} > 0$

3.3.3 Thermal comfort and sensations

Although there are various types of thermal comfort indices which was covered in the literature review, for the analysis done in this report, only 3 will be used, namely:

1. PMV and PPD: This is widely used in the literature for measuring comfort and sensations
2. Local equivalent temperatures and ISO comfort indices
3. Zhang's thermal comfort prediction

Preference will be given to Zhang's thermal comfort prediction as it is one of the newer comfort models with a more comprehensive comfort measurement for localised cooling. Extensive research was done with the cooling of each individual body parts to see how the subject responses to the cooling in terms of comfort.

As it has extensive data on the comfort level with the cooling of certain body parts, it is more suitable for personalised ventilation systems.

3.3.3.1 PMV and PPD

The predicted mean vote otherwise known as PMV uses the following in order to determine the value of PMV:

1. Global thermal resistance of clothing, R_{cl}
2. Metabolic rate, M
3. External work done, W
4. Mean air temperature, T_{air}
5. Mean radiant temperature, T_{rad}
6. Relative air velocity, v_{air}
7. Partial water vapour pressure of adjacent air, p_{air}

Note that variables which describe the properties of the manikin, such as temperature of the skin, are not used for PMV as mentioned in the literature review. The calculation of PMV uses the average of the above variables weighted by the surface area, A_i of the manikin it affects.

$$x = \frac{\sum_i x_i \cdot A_i}{\sum_i A_i} \quad (3-96)$$

The PMV should only use when the environmental conditions are as follows:

- M range from 46 to 232 W/m^2

- T_{air} range from 10 to 30°C
- T_{rad} range from 10 to 40°C
- v_{air} range from 0 to 1 m/s
- p_{air} range from 0 to 2700 Pa (RH between 30 to 70%)
- l_{cl} range from 0 to 2 clo

The PMV index is analogous to the difference between the actual heat flow from the body in a given environment compared to the heat flow required for an optimal thermal comfort condition. This is given by:

$$PMV = [0.303 \cdot \exp(-0.036 \cdot M) + 0.028] \cdot L \quad (3-97)$$

where L is the thermal heat load on the body. In other words, this is considered the difference between the internal heat being produced by the body and the sum of all the heat losses to ambient environment for a person in a perfect comfort.

$$L = (M - W) - \{3.96 \cdot 10^{-8} \cdot f_{cl} \cdot [(T_{cl} + 273)^4 - (T_{rad} + 273)^4] + f_{cl} \cdot h_c \cdot (T_{cl} - T_{air}) + 3.05 \cdot 10^{-3} \cdot [5733 - 6.99 \cdot (M - W) - p_{air}] + \max(0.42 \cdot [(M - W) - 58.15], 0.0) + 3.53 + 1.7 \cdot 10^{-5} \cdot M \cdot (5867 - p_{air}) + 0.0014 \cdot M \cdot (34 - T_{air})\} \quad (3-98)$$

where

- f_{cl} : Ratio of the person's surface area with clothes compared to the surface area when nude.
- T_{cl} : Global human clothing temperature (calculated through iteration)
- h_{cl} : Convective heat transfer coefficient

f_{cl} is given by:

$$f_{cl} = \begin{cases} 1 + 0.2 \cdot l_{cl} & \text{if } l_{cl} < 0.5 \text{ clo} \\ 1.05 + 0.1 \cdot l_{cl} & \text{if } l_{cl} > 0.5 \text{ clo} \end{cases} \quad (3-99)$$

T_{cl} is given by:

$$T_{cl} = 35.7 - 0.028 \cdot (M - W) - R_{cl} \cdot \{39.6 \cdot 10^{-9} \cdot f_{cl} \cdot [(T_{cl} + 273)^4 - (T_{rad} + 273)^4] + f_{cl} \cdot h_c \cdot (T_{cl} - T_{air})\} \quad (3-100)$$

h_c is given by:

$$h_c = \max[2.38 \cdot (T_{cl} - T_{air})^{0.25}, 12.1 \cdot \sqrt{v_{air}}] \quad (3-101)$$

With the PMV formula will give a comfort value from -3 to +3 related to the ASHRAE-scale. The PMV can then be used to find the percentage of person's dissatisfied, PPD.

$$PPD = 100 - 95 \cdot e^{-0.03353 \cdot PMV^4 + 0.2179 \cdot PMV^2} \quad (3-102)$$

3.3.3.2 Local Equivalent Temperatures and ISO comfort indices

This is based on the ISO 14505-2 for assessment of thermal conditions in an automobile. This is catered to be used for asymmetric thermal conditions on a person.

The definition of the equivalent temperature, T_{eq} is the temperature of a homogenous room with a radiation background temperature equal to the room's air temperature with zero air speed. The person will experience the same heat loss by convection and radiation in that space as in actual conditions.

In mathematical notation:

$$T_{eq} = T_{rad} = T_{air} \quad (3-103)$$

and for the heat loss of the person in the room as compared to realistic conditions:

$$Q_{actual} = Q_{eq}(T_{eq}) \quad (3-104)$$

This method can be used for both global and local equivalent temperatures. Therefore if Q_{eq} is in the form of a local equivalent temperature of a certain body part:

$$Q_{actual} = A_{sf} \cdot h_{c,mix} \cdot (T_{sf} - T_{eq}) + A_{sf} \cdot h_r \cdot (T_{sf} - T_{eq}) \quad (3-105)$$

where T_{sf} is the outer surface temperature of the body element

And h_r and $h_{c,mix}$ are as follows below.

$$h_r = \sigma \cdot \varepsilon_{sf} \cdot \varepsilon_w \cdot \psi_{sf-w} \cdot (T_{sf}^2 + T_{eq}^2) \cdot (T_{sf} + T_{eq}) \quad (3-106)$$

where $\psi_{sf-w} = 1$ and $\varepsilon_w = 0.9$

$$h_{c,mix} = \sqrt{a_{nat} \cdot \sqrt{|T_{sf} - T_{eq}|} + a_{frc} \cdot v_{a,eff} + a_{mix}} \quad (3-107)$$

where $v_{a,eff} = 0.01 \text{ m/s}$ and a_{nat} , a_{frc} , and a_{mix} are constants that differ based on which manikin sector. These value can be found in (Dusan, 1998).

As for the local thermal comfort indices for ISO 14505-2, this can be found based on the data given in that standard. The data for comfort is obtained by experimental sampling with variations of unsymmetrical thermal loads taking into the analysis. The comfort index is between 1 to 5 where 1 is too cold and 5 being too warm, and 2, 3 and 4 being under the comfortable region.

3.3.3.3 Zhang comfort prediction

Zhang's comfort prediction and thermal sensation index is based on experiments which involved a large sample size involving cooling of different body parts and surveying of the

comfort level. It uses the temperatures of the manikin directly in order to calculate and determine the sensation the person would feel.

This model is the most complex comfort and sensation model available at the moment, depending largely on regression analysis to evaluate the data points. The theory shown here would not be the method in its entirety but just focusing on the main concepts behind this form of analysis.

Zhang's prediction starts off with determining the sensation first based on the sensation index SI_i for each of the body parts. With the SI_i , SI_o the overall thermal sensation can be obtained. Next, with the SI_i and SI_o , the local thermal comfort index, L_c and the overall thermal comfort index C_o can be found.

3.3.3.4 Local Thermal sensation index

With reference to the governing equations for the manikin temperatures, this will be used for determining the SI_i :

$$SI_i = SI_{sta}(\Delta T_{sk}, \Delta T_{sk,m}) + SI_{dyn}(\dot{T}_{sk}, \dot{T}_{hy}) \quad (3-108)$$

where:

For the static portion of the equation, SI_{sta} :

$$SI_{sta} = 4 \left[\frac{2}{1 + e^{-C_1 \Delta T_{sk} - K_1 (\Delta T_{sk} - \Delta T_{sk,m})}} - 1 \right] \quad (3-109)$$

where ΔT_{sk} holds the same definition as the term under the manikin theory in the earlier sections. The coefficients C_1 and K_1 can be found using the Figure 3-3. These 2 coefficient depends on which body part and if the at the moment in time if $\Delta T_{sk} < 0$ or > 0 .

For the dynamic portion of the equation, SI_{dyn} :

$$SI_{dyn} = C_2 \dot{T}_{sk} + C_3 \dot{T}_{hy} \quad (3-110)$$

where likewise to SI_{sta} , C_2 and C_3 are also dependent on the body part and $\dot{T}$ is the rate of change of temperature in (K/s).

3.3.3.5 Overall thermal sensation index

SI_o can be found as a weighted average of the various SI_i of the different body parts.

$$SI_o = \frac{\sum_{bp=1}^{18} W_{bp} SI_{bp}}{\sum_{bp=1}^{18} W_{bp}} \quad (3-111)$$

where weighting factor W_{bp} for each of the body parts can be found using:

$$W_{bp} = a_{bp} (SI_{bp} - Sm) \quad (3-112)$$

where a_{bp} is the coefficient that can be found from Figure 3-3. The weighting factors equation takes into account the local derivation towards the mean sensation Sm .

$$Sm = \frac{\sum_{bp=1}^{18} A_{bp} SI_{bp}}{\sum_{bp=1}^{18} A_{bp}} \quad (3-113)$$

Body Parts	static local sens.			dynamic local sens.			global sensation		local comfort...							
	$\Delta T_{sk}<0$ $\Rightarrow C_1=$	$\Delta T_{sk}\geq 0$ $\Rightarrow C_1=$	K_t	$\dot{T}_{sk}<0$ $\Rightarrow C_2=$	$\dot{T}_{sk}\geq 0$ $\Rightarrow C_2=$	C_3	$S_1<S_m$ $\Rightarrow S=$	$S_2\geq S_m$ $\Rightarrow S=$	C_{31}	C_{32}	C_6	C_{71}	C_{72}	C_8	N	
Head	0.38	1.32	0.18	543	90	0	-0.13	0.21	0	1.39	1.27	0.28	0.4	0.5	2	
Face	0.15	0.7	0.1	37	105	-2289	-0.15	0.3	-0.11	0.11	2.02	0	0.4	0.41	1.5	
Neck	0.4	1.25	0.15	173	217	0	-0.13	0.23	0	0	1.96	0	0	-0.19	1	
Chest	0.25	0.6	0.1	30	196	-2135	-0.23	0.23	-1.15	0	1.86	0.92	0	0	1.5	
Back	0.3	0.7	0.1	60	192	-4064	-0.23	0.24	-0.5	0.59	2.22	0.74	0	0	1	
Pelvis	0.2	0.4	0.15	75	137	-5053	-0.17	0.15	-1	0.38	2.7	0.83	-0.64	-0.75	1	
Up.arm	0.29	0.4	0.1	155	167	0	-0.1	0.14	-0.43	0	2.2	0	0	-0.33	1	
Lo.arm	0.3	0.7	0.1	144	125	0	-0.1	0.14	-1.54	0.34	2.35	1.18	0.28	-0.41	1	
Hand	0.2	0.45	0.15	19	40	0	-0.04	0.04	-0.3	0.8	1.89	0.48	0.48	0	1	
Thigh	0.2	0.29	0.11	151	253	0	-0.13	0.26	0	0	1.95	0	0	0	1	
Lo.leg	0.26	0.4	0.1	208	212	0	-0.13	0.26	-1	1.5	1.27	0.4	1.22	0.36	1.5	
Foot	0.25	0.28	0.15	109	162	0	-0.08	0.14	-2.31	0.21	1.82	0.5	0.3	-0.25	2	

Figure 3-3: Coefficients for Zhang's comfort and sensation models

3.3.3.6 Local thermal comfort index

The formula for the local thermal comfort index is as follows:

$$L_c = \left[\frac{left_{slope} - right_{slope}}{e^{5(SI+offset)} + 1} + right_{slope} \right] \cdot f + L_{c,max} \quad (3-114)$$

where

If $SI_o < 0$,

$$\begin{aligned} offset &= C_8 + C_{31} \cdot |SI_o| \\ L_{c,max} &= C_6 + C_{71} \cdot |SI_o| \end{aligned} \quad (3-115)$$

Else if $SI_o \geq 0$

$$\begin{aligned} offset &= C_8 + C_{32} \cdot |SI_o| \\ L_{c,max} &= C_6 + C_{72} \cdot |SI_o| \end{aligned} \quad (3-116)$$

And

$$left_{slope} = \frac{4 + L_{c,max}}{|-4 + offset|^N}, \quad right_{slope} = \frac{-4 - L_{c,max}}{|4 + offset|^N} \quad (3-117)$$

And

If $SI + offset < 0$,

$$f = -|SI + offset|^N \quad (3-118)$$

Else if $SI + offset \geq 0$,

$$f = |SI + offset|^N \quad (3-119)$$

where for all the above equations, parameters $C_{31}, C_{32}, C_6, C_{71}, C_8$ and N can be found in Figure 3-3.

3.3.3.7 Overall thermal comfort index

The overall comfort index is calculated similarly to the overall sensation index, with the use of the weighted average of the body parts:

$$L_{c,bp} = \frac{\sum_i A_i L_{c,i}}{A_{bp}} \quad (3-120)$$

where $L_{c,i}$ is the local comfort index and A_i is the surface area of a sector of a body part

The overall thermal comfort index is determined using 2 rules depending on the circumstances:

Rule 1

$$C_o = \frac{1}{2} [\min_1(L_{c,bp}) + \min_2(L_{c,bp})] \quad (3-121)$$

The average of the 2 lowest $L_{c,bp}$ will be used to determine the C_o for rule 1

Rule 2

$$C_o = \frac{1}{3} [\min_1(L_{c,bp}) + \min_2(L_{c,bp}) + \max(L_{c,bp})] \quad (3-122)$$

The following rule 2 is only used when all of the following are met:

- The second lowest local comfort vote is > -2.5
- The subject has some control over his/her thermal environment (air conditioning)
- Or the thermal conditions are transient.

For both the rules, if both of the hand or the feet falls under the 2 most uncomfortable body part, the hand or foot which has the 2nd lowest comfort value should be ignore. Instead, the 3rd lowest local comfort vote will be used thus assumed as the 2nd lowest vote.

Therefore in the simulations in this report, usually rule 2 is used, as it is regarded that the user has control over his or her environment and the thermal conditions can be considered to be transient.

4 Simulation of solar loading on a vehicle

4.1 Simulation setup

In this section, four analyses on the heat load of the vehicle will be discussed, while the results will be discussed in section 4.2. The four analyses on heat loading as mentioned in the section 1.3 are:

1. Understanding the changing of the heat load profile throughout the day
2. The different phases in the vehicle heat loads
3. How various methods can help in the heat load control of the vehicle
4. How the change in the vehicle geometry can affect the heat load on the vehicle

Below is surface mesh used for the heat loading simulation. The vehicle model used is based on a Mercedes Benz Vito. This vehicle is similar in size to the electric vehicle that will be constructed by TUM Create, therefore, may serve as a guide to the type of heat loads which will be experienced.

Figure 4-1: Surface mesh of Vito used

Figure 4-2 highlights the dimensions used for the vehicle.

Figure 4-2: Vito dimensions

Theseus FE

The numerical simulation in this section was done using the software, Theseus FE. Theseus FE and Radtherm are software offering similar capabilities, specialise in thermal analysis of solids. Their algorithm and governing equations allow for the simulation of multi-layer solid objects or composites allowing for the analysis of temperature distribution through the layers.

One useful feature of Theseus FE is its ability to numerically cater to layers of materials without the need for physical representation of these materials via finite elements or volumes. This means complex geometries can be represented with just the use of a surface mesh. This allows for large scale simulation of an entire part in an efficient way with reasonable accuracy. This will draw comparisons with CFD solvers like Fluent, which can perform conjugate heat transfer. However, Fluent has no efficient way to cater to composite materials, if all materials are to be resolved, Fluent will need to have the finite element mesh in place. Another option is the use of the shell conduction function. For composite materials, the overall conductivity of the materials will be used instead. There are also other limitations of using Fluent alone for a complete vehicle simulation which would not be covered here.

Theseus FE allows for 3 heat transfer methods, conduction, radiation and convection. As Theseus FE is not a CFD solver, it is only able to cater to forced convection with an user input for velocity. Therefore, a reasonable estimate has to be made to give a reasonably accurate result. It makes use of various analytic functions for calculating of the convection depending on the orientation for free convection.

This software also includes a thermal comfort manikin in the form of a surface mesh. This manikin is based on the Fiala physiological model and has various comfort models such as the University of California, Berkeley (UCB) comfort model and Fanger's PMV and PPD allowing for thermal comfort analysis of the air distribution system. As such, when a simulation is run, the results from these various comfort models can be obtained.

From a simulation perspective, Theseus FE can be coupled with a CFD solver like Fluent. This coupling allows for real time calculations with the use of the 2 software. In simple simulations, the temperature of the surfaces will be transferred from Theseus FE to Fluent and the convection heat transfer coefficient will be transferred over. This operation will be continued until convergence is reached. This way, the simulation time will also be sped up as Theseus FE is able to calculate thermal heat transfer very efficiently, allowing Fluent to solve the fluid flow. As a result, Fluent would not have to perform other calculations such as radiation and conduction heat transfer.

Input parameters

For all the simulations ran, the materials of the vehicle were kept constant. The materials used in simulations are the actual materials used in most vehicles. Theseus FE has a database of such materials. Both long range and short range wavelengths radiation models were switched on and used to consider radiation heat transfer.

Convection heat transfer was considered for both the cabin and external surfaces. As stated above, the forced convection will have the velocity term assumed for the calculation of the Reynold's number and thus the Nusselt number for determining the heat transfer coefficient. Conduction through the material will be calculated depending on the material thickness and conductivity. Transient simulations will use the thermal capacity of the material. These properties are defined for all the materials used.

A basic form of ventilation function is used to simulate how the air conditioning would behave. Ventilation functions such as having the cabin temperature as a fixed input or having an evaporator performance chart will allow for the different studies to be made.

The solar radiation with the intensity of the radiation i.e. diffused and direct, the solar altitude and azimuth are taken into consideration. For most cases, a clear sky scenario is taken into consideration. Ambient temperature and the sky temperature are also inputs in the simulation.

4.1.1 Changing of the heat load profile throughout the day

This case is conducted to allow for the understanding of how the heat load might occur in a car if it was driven throughout the day. This information is of importance as it gives an estimate when the peak heat loads might occur and what such a heat load might be.

This angle which resulted in the peak heat load will be used as the sun's angle and strength for the simulation of the other cases. This case is simulated to obtain how the sun will behave and affect the cabin heat load in a period of 13 hours.

Using the Vito CAD model, details of the sun's movement, angle and intensity, ambient temperature over the 13 hours period, the simulation was ran with the assumption that the vehicle is moving at 30km/h, which alsomeans a velocity of 30 km/h is assumed over the external of the vehicle. The internal surfaces of the cabin are assumed to have an air velocity of 0.15 m/s.

The solar declination and angle are as follows, noting that the solar angle resolution is 1 hour:

Time (s)	Ambient temperature (°C)	Sky Temperature (°C)	Direct Radiation (W/m^2)	Diffused radiation (W/m^2)	Solar azi (°)	Solar alti (°)
0	27	21	0	0	0	0
3600	27	21	14.741	0.855	97.37948	1.718873
7200	28	22	750.41	43.523	98.06703	16.7
10800	28	22	937.4596	54.372	99.47384	31.52241
14400	30	24	1010.522	58.61	102.1121	46.26061
18000	31	25	1045.289	60.6268	107.7213	60.79082
21600	32	26	1061.396	61.561	124.1206	74.40315
25200	33	27	1065.311	61.788	193.0722	81.05823
28800	33	27	1058.388	61.38653	242.6305	70.90639
32400	33	27	1038.176	60.21423	254.3046	56.87923
36000	32	26	995.9543	57.76535	258.7857	42.28142
39600	32	26	904.503	52.4612	260.9921	27.514

43200	31	25	644.03	37.35378	262.1604	12.67783
46800	30	24	0	0	270	0

Table 2: Radiation values throughout the day

The solar radiation for direct and diffused radiation is calculated using the equations for heat load from the ASHRAE handbook. This method is known to have quite a large error of about 41% for diffused radiation (Wong & Chow, 2001). However, this is not a significant problem as diffused radiation affecting the heat load tends to be only a small percentage of the overall radiation.

Three simulations were carried out each with a different orientation of the vehicle with respect to the south direction at a declination of 10° . Two other cases were run with another two solar declinations of 0° and 20° . This is to understand how the heat load will differ with standard vehicle geometry at different orientations. With this understanding, the other simulations will also be conducted using the same vehicle orientation.

Figure 4-3: The solar declination and the sun's route over the car

Thus, the scenario of this simulation can be taken as a vehicle driving for 13 hours without stopping, from sunrise to sunset. This is simulated to be moving only in a straight line in one direction with respect to the south direction.

4.1.2 Heat loads at the different phases of vehicle operation

For every vehicle, there are three basic types of air conditioning and heat loading every vehicle will go through. This would be:

- **Parking-** This is when the car is parked. This scenario is the worst when the vehicle is parked under the sun. In ordinary vehicles, this result in a build-up of thermal energy in the vehicle cabin as the trapped heat cannot escape. Temperature in the vehicle can reach very high temperatures, possibly up to $60-70^\circ\text{C}$. The heat load will increase until it reaches a steady state. This means the heat coming into the vehicle is at equilibrium with the heat leaving the vehicle. The air conditioning is not running at the moment, so the air conditioning load is zero.
- **Cooling down-** This is when the vehicle is about to move off and the engine has just been started. The air conditioning would have just been switched on and would have

to start with the heat removal process. Due to the intense heat build-up during parking, the load of the air conditioning would be the highest. This term for the air conditioning would be called the pull down. The air conditioning would need to cool the air, the surfaces and the materials such as the seat and so forth. Therefore, if the materials have a high thermal capacity, the cooling capacity needed will be increased. The more powerful the air conditioning is ie, having high mass flow rate and low temperature inlet, the faster the cabin would be cooled. As such, having a high capacity air conditioning would be vital for this aspect of the cooling.

- Steady state- this stage is when the cabin has been cooled and the vehicle is in operation. The air conditioning load have been stabilised and should no longer be working at full load.

This will give a complete picture of what is expected of the air conditioning and how the cabin conditions will respond. It would be important for the air conditioning system designers to ensure that the air conditioning system can perform within this operating range.

For this simulation, the parking and the pulling down scenario will be simulated in a continuous process. The scenario will simulate parking in the sun for 2 hours and 15 mins, followed by the cooling of the space till steady state. For parking, the free convection will be assumed for both the air in the cabin and external surfaces.

In particular to this case, the air blowing into the inlet is fixed at $0.15 \text{ m}^3/\text{s}$ and the temperature follows a linear relationship with the incoming recirculated air from the cabin. In other words, similar to how an evaporator operates, the incoming air of a certain temperature will be cooled to a respective temperature following this type of relationship.

4.1.3 How various methods can help in the heat load control of the vehicle

One minor aspect of lowering the heat load of the vehicle is the use of passive thermal control measures. This simulation will give a rough estimate how such methods might lower the amount of heat entering the vehicle and thus lowering the cooling capacity needed. Some methods that will be discussed are the use of:

- Window covers when parking
- Ventilation system to blow in air from the ambient to cool the interior
- Mesh seats
- Shiny reflective materials

There are other types of methods that can assist in lowering the heat load such as changing of colours of the interior. However, this will not be covered here.

The solar input will be identical to the other simulations. The only change will be the properties of the respective surfaces. The convective heat load will be compared with other cases. Note that some of the methods shown above only affect the vehicle significantly in certain scenarios. Such as the window covers will only help during parking and not during the cooling down or steady state.

4.1.4 How the change in the vehicle geometry can affect the heat load on the vehicle

The following geometry of the windscreen and the side windows were adjusted to understand how the geometry can affect the heat load on the vehicle. The main heat source that is to be avoided would be the solar. Obviously, having an angle that curves in providing an overhang will offer the best protection against direct and diffused solar radiation as no direct radiation would be able to enter the vehicle.

Figure 4-4: Windscreen and side windows geometry changes

The angles will be adjusted between 70° to -30° from vertical for the windscreen and between -12° to 14° for the side windows. The windows sizes, in particular the first window are adjusted so that the border of the windows is not unnaturally thick. Again for this case, the solar radiation used will be the peak radiation which should result in the peak heat load.

The important part to note about such changes in geometry is that such change in angles can affect the aerodynamics of the vehicle. However the aerodynamics of the vehicle will not be considered in this report. The aim here is to present the amount of improvements such designs will have compared to the original vehicle design.

4.1.5 Windows Properties

It will be shown in the results section that windows are the biggest factor that allows heat going into the vehicle. As such, if no physical obstructions are used to block the solar radiation from entering the vehicle, the next best solution would be looking into better window technologies.

One aspect that would be interesting to know is how the radiation properties of the window can affect the heat load in the vehicle. The absorptivity, transmissivity and reflectivity will be adjusted to highlight the amount of differences a combination of 2 of these factors can have. This would assist material engineers doing research in glasses to understand how best to

optimize the glass material in blocking sun radiation. The windows radiation properties were adjusted in the following combinations.

Group	A	B	C	D	E	F	G	H	I	J	K	L	M	N	O
absorptivity	1	0.8	0.6	0.4	0.2	0	0	0	0	0	0	0.2	0.4	0.6	0.8
reflectivity	0	0.2	0.4	0.6	0.8	1	0.8	0.6	0.4	0.2	0	0	0	0	0
transmissivity	0	0	0	0	0	0	0.2	0.4	0.6	0.8	1	0.8	0.6	0.4	0.2

Table 3: Window radiation properties

4.2 Results and Discussion

4.2.1 Changing of the heat load profile throughout the day

This simulation is to calculate, in theory, the highest heat load in the vehicle. It will give a conservative estimate for the calculation of the air conditioning loading on the electric vehicle battery pack and how this might affect the range.

The simulation was run with the solar radiation for 13 hours from sunrise to sunset, keeping a declination of 10° .

Figure 4-5: Thermal radiation loading at different time and orientation

This simulation does cater to the thermal storage effect, but it is not pronounced in this simulation. It would be clearly seen in other simulations. This is due to two reasons. Firstly, solar radiation going into the cabin is high resulting in fast heating. Secondly, besides the seats, the other materials such as insulations and so forth are not as dense and thick as concrete in buildings. This results in a much shorter response for the air conditioning to react to the heat load from convection. The heat load from convection is an important property, as this is the heat load that would be transferred to the air. Only when the heat is transferred to the air, energy can be removed by the air conditioning system.

What can be seen from Figure 4-5, is an initial sharp peak almost immediately after the calculation starts. This is due to the high initial temperature set at 33°C for all surfaces in the vehicle. After the surfaces have been stabilised for that time period, the convection load gradually increases till it reaches a peak under high heat load at about 1000-1300 hours.

The peak load occurs when the vehicle is facing directly south. When compared to the solar radiation load, although the solar radiation only peaks at 1300 hours, the convection load only peaks at about 1500 hours. A small part of this can be attributed to the thermal storage effect, but a bigger contribution would be from the angle of the sun rays penetrating the vehicle windows at that hour. A larger amount of sunrays was able to enter the vehicle as the side windows were also utilised instead of just the windscreen at the earlier hours. This can

be seen in Figure 4-6, how the solar radiation enters the cabin area. Notice that at 3pm, although the solar radiation is not as high, a bigger surface area of the vehicle is exposed to the sun. This results in a larger amount of solar radiation energy being transmitted through the windows and absorbed by the areas exposed to the sun.

Figure 4-6: Solar radiation comparison, left at 1pm and right at 3pm

When the vehicle is positioned at 90° to the south, the solar radiation entering the cabin will peak at about the same time the solar radiation peaks. This can be attributed to the side windows and windscreen being the main source of radiation. This is due to the extreme inclined angle of the windscreen to the sun. As such, as the vehicle turns from 0° to 90° , it can be seen that the heat load graph changes from a graph with 1 peak to 2 peaks. The overall heat load will also increase.

From the results, it gives an idea how the heat load will behave for this extreme scenario. In reality, the vehicle should not be exposed to such extreme heat due to shading from buildings, trees and cloud cover. In Singapore's landscape, this would be an important aspect to consider when realistic air conditioning loading is considered. It is also unlikely that the vehicle will be running a straight line facing south all the way.

Although not shown, the declination of the sun affected the heat loading although the peak was the 10° case (shown). The peak load for the 0° and 20° cases resulted in a peak load drop of 1.4% to 1.5% which is not very significant. Using the angle of the sun and the vehicle, along with the solar radiation intensity of this peak loading, this information will be taken as the radiation heat input for other simulation cases. Do note that there are no occupants in the vehicle. The peak convective heat load for this case is 1788.3W. The heat load can increase by 100-130 W per person. Moreover, the calculation has not catered for humidity.

4.2.2 Heat loads at the different phases of vehicle operation

With the peak solar heat load determined, the next phase would be to understand how head load in the vehicle changes during the three major phases namely, parking, cool down and steady state.

The temperature of the cabin air and the convection load to the air is shown in Figure 4-7. The convection load present only starts from 2 hour and 15 minutes as the air conditioning only starts operation when the cooling down phase starts.

Figure 4-7: Temperature and convection loading

The results show that the cabin temperature peaks at about 63°C after a descending increase of temperature rise. This shows how extreme temperatures in the cabin can get. After the air conditioning is switched on with a constant flow rate of 0.150 m³/s, it is shown that the cabin temperature drops from quite rapidly over a short span of 25 minutes to a more comfortable 23°C. The temperature reaches a steady state of about 18°C.

Meanwhile, the convection load will increase up to about 3100 W. This is due to the cool air entering the cabin space inducing a higher convection rate because of the larger temperature difference between the air and the surfaces. As the surface cools and reaches equilibrium, the convection cools and reaches steady state of 2300 W.

Figure 4-8: Steady state convection load

The steady state convection load for a cabin temperature of 23°C was found to be 1800W. This value will be compared for other geometry in a later section to understand how the convection load is affected.

Understanding what are the sources of heat that would be transferred to the cabin air, would allow for better heat removal and if possible promote energy savings and improved thermal comfort. Note that the simulation uses a more updated model as the simulation was updated with the new vehicle size.

The simulation gives thermal and heat transfer information for each of the surfaces. Convection and radiation are the most important heat transfer processes what will affect air conditioning heat load and the comfort respectively. The Figure 4-9 shown below gives an idea of the amount of energy transferred to the air via convection.

Figure 4-9: Convection heat load when cabin temperature of 21°C

The results show that the 3 largest source of heat were from the windscreen and windows marked in yellow. In addition, to that is the top surface of the dashboard and the floor. It was found that usually the surfaces which have been exposed to solar radiation thus result in having high temperatures, providing the heat gains into the cabin. It would be interesting to note that the convective heat load shown is not on a per area basis but the total heat load from a particular surface. Even though the roof is exposed to the sun directly overhead, the heat gain from convection lower compared to the windows. This shows that the insulation in the roof is quite effective in preventing heat transfer in to the cabin.

The second figure shows the amount of radiation absorbed by each surface. Similarly, the hotter surfaces such as the top of the dashboard and windows were the main source of radiation emitters. Unlike the windows, the windscreen is shown to have a very low amount of emissions or absorptions. This can be attributed to a large amount of energy being radiated from the dashboard into the windscreen resulting to a lower radiation being emitted from the windscreen.

Figure 4-10: Absorbed radiation from surfaces at 21°C cabin temperature

4.2.3 How various methods can help in the heat load control of the vehicle

This section shows the energy savings method at a certain operational phase can that affect the convection heat loading on the vehicle significantly.

Windows cover when parking

The windows cover can be used when during parking. This is to prevent the radiation from being transmitted into the cabin area. As transmission of the radiation through the windows is the largest source of heat gain, this would help in reducing the energy required during the cool down phase.

The simulation showed that after 2 hours and 15 minutes, the cabin reaches 53°C compared to the original cabin with windows of 63°C. Due to the lower temperature, it was found that the total energy difference used by the air conditioning is quite substantial. The original loading of the air conditioning by convection is 2.5 kWh while with the window covers, it gave a load of 1.83 kWh. The energy saved was about 27% lower.

Ventilation system to blow in air form the ambient to cool the interior

The ventilation system pushes ambient air into the cabin during parking as an attempt to keep the temperatures in the cabin cool. Similar to the window covers, this solution will help the air conditioning to save the energy required during the cooling down phase.

With a flow rate of 0.05 m³/s and a presumed air velocity of 0.05 m/s, the temperature reached after 2 hours 15 minutes is 48°C. The energy saved with such an arrangement is found to be 27% lower than the original with only 1.82 kWh.

This method if used with the windows binds or cover will allow a much cooler cabin during parking. If fact, forced ventilation might not be necessary, as long airflow via natural

convection is promoted, a substantial amount of energy can be saved. This is due to the constant source of cooler air flowing into the cabin bringing the temperature of the cabin surfaces down. In the usual cabin designs, the air cannot escape. This results in the air temperature rising until the heat transfer coming into the cabin is equal to the heat transfer leaving the cabin.

Mesh Seats

Seats with little or no foam have a lower mass, and thus a lower thermal capacity. This can be considered important when a vehicle is getting soak during parking. When the vehicle is parked, solar radiation will be providing thermal energy to heat up the cabin. As the radiation is absorbed and stored in the material, the temperature of the material will rise. Although some materials have very high thermal capacity, resulting in a slow increase in temperature over time, parking of the vehicle can occur for hours. This gives ample time for most materials store a large amount of energy until the heat exchange between the environment and the vehicle reaches equilibrium.

Any amount of energy increase of the material in the cabin will result in need of the removal of the energy from the cabin by the air conditioning system. Therefore, having materials with a large mass and with a high thermal capacity will result in a lot more energy removal for air conditioning.

One simulation ran involved changing the seats from the convectional type with a foam core compared to the other type with just a fabric alone. The latter is to simulate the mesh type of seats with little or no stuffed foam.

The results showed quite a large drop in energy requirement for cooling, with about 46% savings. This is because in the simulation, the seats are the items in the cabin with the most mass. By removing the foam in the seats, the thermal capacity was reduced by 97%. In numerical terms, the total amount of energy needed to cool convectional seats were 2.5 *kWh* while the ultra-thin seats needed only 1.35 *kWh*. This also lightens the vehicle lowering the inertia, saving energy.

Shiny reflective material

The other solutions covered above have only lead to energy savings for a vehicle during cooling down. If the vehicle has been cooled, and the temperature and convective load has reached a steady state, the above solutions would have little or no effect. The main reason would be that they don't reduce the amount of radiation entering the cabin.

Steady state conditions in the cabin will need the direct reduction of the heat load entering the cabin. One way would be the use of a shiny reflective material as the vehicle body to reduce the absorption of the radiation and increase the amount reflected. The material tested for this case was to compare the normal painted vehicle with one using polished aluminium as the car body surface.

Even when the surface was reflective, the drop in the heat load into the vehicle was not very significant. There was only a 6% decrease of convective heat load, at steady state. Comparing

the steady state load of 1814.24W for a convectional painted vehicle, the shiny surfaced vehicle had a convection load of 1696.34W. This meant that the existing insulation between the layers of the outdoor and indoor surfaces is quite effective for minimising heat transfer for the opaque surfaces.

4.2.4 How the change in the vehicle geometry can affect the heat load on the vehicle

Since solar radiation plays such a huge effect on the heat load in the vehicle, one attempt to minimise that was with the use of the vehicle geometry itself. Therefore, the geometry of the vehicle and the effectiveness of blocking out the radiation were explored.

As shown in section 4.1.4, the geometry was varied from -30° to 70° for the windscreen and -12 to 14° for the side windows, taking reference from the vertical plane. The following results were obtained:

Figure 4-11: Windscreen convective heat load and % improvement compared to the base model

57° is the windscreen angle of the base model. One can see how the percentage improvement increase as the angle decreases towards 0° . In fact, up to 40° , the improvement increased quite drastically. After 40° , the improvement in the convective heat load becomes lesser. This can be attributed to the drop of radiation exposure on the cabin floor and seats. As the radiation become more restricted to only the dashboard, the convective heat load drops more significantly. Since a compromise have to be made between lower thermal load and the aerodynamics or style of the vehicle, extreme windscreen angles would not be ideal. Therefore, 40° would be able to give a quite a significant improvement compared to the base model.

The Figure 4-12 shows how the angle of side windows affects the convective heat load of the cabin. It can be immediately noticed that the decrease in heat load is linear. This is unlike the

windscreen which has a larger impact on how the solar radiation is exposed on the surfaces of the cabin leading to larger improvements. Moreover, side windows are not steep to start with and therefore, making it less steep to minimise solar radiation would not lead to large improvements. As such side windows are not a real priority for study to lessen solar radiation.

Figure 4-12: Side windows convective heat load and percentage improvement

4.2.5 Windows properties

Windows is the main source of heat energy coming into the cabin via solar radiation. It was important to understand how radiation properties like absorptivity, transmissivity and reflectivity can affect the cabin thermal loads.

Using the convective load to the cabin as a benchmark, the following results were found:

Figure 4-13: Convective heat load for each absorptivity, transmissivity and reflectivity combinations

Group F to K varies with increasing transmissivity and decreasing reflectivity. This shows that high transmissivity will lead to the highest convection load, which is common knowledge.

What is interesting is how much the convective heat load jumps comparing group F and group G. The difference between group F and G is that transmissivity was increased from 0 to 0.2 and reflectivity dropped from 1 to 0.8. This results in a convection load increase of almost 67%.

This means that with windows with a capability of filtering the solar IR spectrum would be very effective. A more conventional way would be to have physical barriers to be preventing direct solar radiation from reaching the windows. This can be with the use of external blinds. However any external devices should be analysed for drag even at low speeds to understand its significance.

5 Simulation of the PV Seat

5.1 CFD Co-Simulation

The objective of the series of simulations is to understand the performance and the advantages of using personal ventilation in the vehicle. The simulations will involve the sun as the main source of radiation and the thermal comfort would be the benchmark for comparison between the conventional air conditioning and the PV type. An energy analysis would also be done to fully understand the benefits of such a system.

The PV design will involve a new seat design that channels the air from the overhead of the person. The back curved portion aims to maintain a certain pressure in the upper body portion of the space preventing the slow moving air from moving away from its original route (downwards to the body). Due to the heat cause by the sun's radiation on the dashboard, the buoyancy effect of the air above the dashboard will push the cool air towards the back of the vehicle. This is most profound during the initial designs where there is no obstacle in air flow to maintain a pressure restricting the movement buoyant warmer air, thus ensuring that the cold air from the air inlet is able to move in its original route.

5.1.1 Tools

2 types of simulation programs were used. Ansys workbench with CFX was used in the independence mesh study and used to calibrate the simulation with the recommended mathematical models for turbulence and energy transfer. StarCCM was used with Theseus FE to provide the flow simulation along with the thermal comfort models. The mesh/element size used by StarCCM is finer than the one used in the mesh independence study.

The meshes were made by Ansys workbench meshing. This program is able to automate the meshing for complex geometrics such as for a vehicle in this case. The settings of the mesh can be control quite explicitly with the use of element sizes, sizing around curvatures and creating of inflation layers and so forth. Due to the complex geometrics, the average element quality is still quite high at about 0.7. The elements numbered about 7 million for the CFD solver. For the surface solver (Theseus FE) the surface elements were lesser.

A mesh with good quality with a mesh size of 4613155 was used in StarCCM to ensure that the simulation would be accurate. However too fine a mesh size would have resulted in too long a computation time and if the mesh is not fine enough, the mesh generated will be of too low quality. This was a practical consideration which we had to consider. Due to the simulation time required and the inherent natural of a coupled simulation, the residual errors are a little higher than usual as mentioned in the next section.

A StarCCM and Theseus FE coupling was done to allow for a coupled simulation between the airflow and the thermal comfort model. Theseus FE is specialised for heat transfers via the 3 main heat transfer mechanisms, namely conduction, convection and radiation. It is able to simulate a radiation heat load like the sun and allow the specification of material layers in the body. This is particularly useful as it allow the material layers in an object to be specified. Most importantly, is the availability of a thermal comfort model which is specialised for localised cooling effects on the human body. It allows for comfort and sensation to be

measured and studied when each part of the human body is exposed to an airflow with certain variables like velocity, temperature and humidity.

5.1.2 Cases to be simulated

5.1.2.1 Calibration

For the first set of cases, this involves the calibration of the mesh size and models. However as the mathematical models used will be based on the simulation and experiment done by Loomans (Loomans, 1998) , no study will be done in this aspect. Moreover the CFD study of low velocity air flow is quite established, with the ideal turbulence models and type of solvers to be used.

The mesh independence study for this simulation was done based on the experiment Loomans conducted for indoor flow, a physical experiment was conducted with a room bring setup with a certain layout and a form of PV air conditioning was simulated.

A total of 20 points from the plan view was taken. Each of these 20 points from the plan view, about 2-5 points of measure were taken for each of these points. These points were used as the points of measurement in the room for temperature and velocity. The magnitude and position of the readings will be used to compare to results of the simulation, in order to ensure that the readings from the simulation are accurate.

The different mesh sizes will also show the trend of the values, how the values differ from mesh to mesh. This will allow for the selection of a mesh size that would be most appropriate for the simulation, and yet not too computationally expensive.

5.1.2.2 PV seat air conditioning vs conventional vehicle air conditioning

The aim of the series of cases is to test how the comfort and energy consumption will be like for the conventional airflow vs the PV seat type of air flow. The CFD flow results will give an idea how the design might work under heavy solar load with one person. The driver, who would be the person which is most exposed to the sun will be simulated. The exposure would be due to the windscreen which offers less protection compared to the side windows.

The PV seat air conditioning will be simulated at slightly different speeds. The volumetric airflow will be matched between the 2 air conditioning types to give a benchmark for comparison.

A few simulations which give rise to the current design would be presented too. Most importantly, the major flaws of initial designs would be explained. This will give some justification to the necessity of the current design.

5.2 Calibration Case

The calibration case is meant to assess if the turbulence models used is accurate in simulation. Most importantly this would also be used to achieve mesh independence. This means that when the mesh gets any finer, the results of the simulation will not differ.

5.2.1 Case setup and boundary conditions

The calibration case consists of a room with one person with a computer. The room size is $(5.16 \times 3.60 \times 2.50) \text{ m}^3$. It consists of an air exhaust at the south east corner of the room, a series of air inlets making the PV type of ventilation system located under the desk. There are two computers, one located at the west side of the room and the other in the middle. There are three lighting fixtures found on the ceiling of the room, and one beside the table.

Figure 5-1: Simulated space with heat loading

1	2	6
	3	
	4	
	5	

Figure 5-2: Velocities at various inlet segments under the table

The boundary conditions are as follows:

Location	Type of boundary	Value
Inlet 1	Velocity inlet	0.2 m/s
Inlet 2	Velocity inlet	0.23 m/s
Inlet 3	Velocity inlet	0.13 m/s
Inlet 4	Velocity inlet	0.11 m/s
Inlet 5	Velocity inlet	0.1 m/s
Inlet 6	Velocity inlet	0.2 m/s
Outlet	Pressure outlet	0.0 Pa
Person, Head	Heat flux	93.8 W/m ²
Person, Chest	Heat flux	36.04 W/m ²
Person, Arms	Heat flux	24.4 W/m ²
Person, Legs	Heat flux	34.86 W/m ²
Computer 1	Heat flux	76.9 W/m ²
Computer 2	Heat flux	76.9 W/m ²
Light_1	Heat flux	72.4 W/m ²
Light_2	Heat flux	72.4 W/m ²
Light_3	Heat flux	72.4 W/m ²
Ceiling	Static temperature	295.45 K
Chair	Adiabatic surface	-
Desk lamp	Heat flux	77.57 W/m ²
North wall	Static temperature	296.35 K
South wall	Static temperature	295.95 K
West wall	Static temperature	295.75 K
East wall	Static temperature	295.85 K
Floor	Static temperature	295.35 K
Table	Adiabatic surface	-
Table stand	Adiabatic surface	-

The above case was simulated using the following mathematical models:

Models	Details
Turbulence	Realisable $K - \varepsilon$ (Two layer wall function)
Buoyancy	On
Gravity	On (for buoyancy)
Solver	Segregated flow energy and momentum
Radiation Off	Off
Gas	Ideal gas
Surfaces	Non-slip
Gas type	Incompressible
Wall function	For all y+

Radiation was not used in this calibration simulation is due to the reason that the temperatures used for the surfaces were of the same value as the experiment. This meant that the temperatures in the experiment already had the influence of the radiation effects. Moreover most of the surfaces in the simulation have a constant temperature boundary

condition or a heat flux boundary condition, except for 2 minor surfaces. This meant that the radiation model had minimal effects for these surfaces.

The dominant flow phenomenon is low speed air flow which can be considered to be incompressible. Although the Reynold's number would be low, the air in the space would be turbulent. This is especially so at the inlet vents. Moreover, for some areas, buoyancy will also play a big part in the air flow patterns in particular in the presence of a heat source.

Based on the research done by Loomans, the same room layout and heat sources were positioned in the room. Temperature sensors and anemometers to measure the air velocity were located in various positions in the room to measure the air properties. These measurements were obtained and compared with the simulation results to verify the accuracy and limitation of the mathematical models used.

Moreover, a series of simulations with varying mesh sizes were tested to ensure that mesh independence could be achieved. Therefore the same case with differing mesh sizes were simulated to see how fine a mesh size was required before the results obtained were not affected by the mesh.

All meshes were created with 3 layer inflation and the element size was controlled. The 3D CAD file was loaded into ANSYS workbench, using a proximity and curvature priority method, and adjusting the minimum and maximum mesh sizes, to generate a range of mesh sizes. The mesh sizes range from 1.6 million to 7.3 million elements. The element sizes range from 0.04 to 0.12 respectively. In addition, some of the CAD geometry was adjusted to allow for better quality meshes. The mesh quality averaged at about 0.8 with a very small percentage of the elements falling to the minimum of about 0.06.

With the boundary conditions given, the results from the simulation were compared to the experiment data for velocity and temperature, and each of the simulated cases.

5.2.2 Results

The simulation at the different mesh sizes showed that the mesh independent was gained at a minimum mesh size of about 0.06, giving a total of about 3111390 elements. The pattern of the air flow showed little difference when compared to a finer mesh of a max element size of 0.04. The temperatures and velocity of the air between the 2 mesh types were very close with many cases, the percentage difference between the 2 is insignificant. However in a few of the cases, the percentage difference between the 2 meshes for either the temperature or velocity was no more than 1%. This will serve as the reference for the rest of the simulations which involves indoor flow for incompressible air.

The results comparing the simulation with the experiment show quite a good correlation between the two. Most of the values of temperature and velocity were within the range of the experiment. Poor correlation can be seen for a few of the values and this can be attributed to the physical phenomena simulated and the characteristics of the flow. This would be discussed in more detail in the following section.

The results also show that the mathematical models used can simulate quite accurately the physics or nature of the flow. The calibration also helps to validate the use of the turbulence

model for use during this simulation, realisable $k - \varepsilon$. Based on the results and mentioned above, there is a possibility the turbulence models might not be as accurate in simulating the air flow for natural convection or buoyancy flow. There is a bigger disparity between the simulated and measured results during those areas where such flow is dominant.

The experiment gave the lower and upper values of velocity and temperature obtained by the measuring devices. The full results list can be found in Appendix C.

5.2.3 Discussion

Based on the results obtained by the simulation, it can be seen that the grid of element size 0.06 is able to achieve mesh independence; elements of smaller size would be able to obtain very similar results. However the important thing to note is that although the mesh size might be suitable for this geometry, on more complex geometry a finer mesh might be necessary to obtain a better quality mesh especially in tight spaces.

In addition, comparing the results with the experimental readings from the anemometer and thermal sensors, it can be shown that the simulation is able to give a very good guide for the temperature readings with most of temperature readings within the experimental range. As for the velocity, the same can be said.

However, the simulation has a few issues with the prediction of free convection air streams. Although the contours lines from the experiment give similar contour lines, the readings from the anemometers showed that the simulation was unable to predict the velocity at the location accurately. The simulation gives a higher velocity for a few of the regions. It was found that these regions were natural convection or buoyant flow dominant. The percentage error between experimental and simulated velocity ranged from 30% to 50%. This is due to the velocity effects in the simulation over predicts the buoyant effects. This might be due to the simulation predicting a vertical flow upwards; however in reality the flow is slanted, leading to higher velocity along a vertical line for the simulation.

The differences between temperature obtained from the experiment and simulation are small though, with percentage errors of about 0.1%. There are the experiment has little issues in predicting the temperature of the air space.

The turbulence models would be deemed suitable as the results obtained by the simulation is accurate with only few portions with more significant error. However, these errors would be due to the airflow direction and pattern and not due to the turbulence model not able to simulate such a phenomenon accurately. Moreover Realisable $k - \varepsilon$ turbulence model, or $k - \varepsilon$ models, is widely used for indoor simulations and has been shown to be able to give good results (Zhai et al., 2007).

This would not be a big issue for the simulation of the PV ventilation cooling in the vehicle. The PV ventilation system is mean to move the air faster against any possible buoyant air flows. Thus, even if the buoyant flow is slightly inaccurate, the result with is dominated by forced convections still relatively accurate as shown in simulation.

5.3 Design methodology of PV seat

In this section, the design process of the vehicle cooling system will be discussed, showing the limitations of earlier designs and the ducting requirements. The PV system aims to cool each individual from the top, in a top down approach. This will allow for each person to have a self-contained climate zone with its own individual air space with temperature.

This will be useful in certain transportation concepts such as car pool, a shared cab or even buses and planes where individuals who might not know each other are sharing a vehicle together. This would minimise the contamination or smells from each person, in addition to allowing each person to adjust the temperature and speed to their own liking. This can vastly improve comfort in the cab.

For all the designs, the target sum of the mass flow rate was about 0.1 kg/s for all the inlets to the cabin airspace. Therefore the designs will involve optimising the velocity and temperatures to meet the heat loading and air flows requirement.

In this section, the airflow is studied based on fixed temperature surfaces obtained from a separate analysis by Theseus analysing just the heat loads. As such the temperatures of the surfaces and the CFD airflow is not synced. However this is suitable as a preliminary analysis, to observe how the air patterns and flow in within the cabin space is like.

The target type of airflow would be the kind which allows for a contained airspace around the person, this would mean little visible air mixing. More importantly, the air flow should be able to overcome the natural convection currents due to high temperature surface caused by the solar loading. In fact for a top down approach in cooling, this is quite an important factor. If the airflow momentum is not high enough and the temperature is low, the warmer air from the hot surface would push the cooler air away and upwards. Thus, the airflow parameters at inlet and the inlet locate is critical for the PV design.

However if the velocity is too high, this may create discomfort. Velocities of up to 0.3 m/s would be considered to be unnoticeable or would not affect the comfort of the person due to velocity. Anything about 0.3 m/s can be felt. The target velocity directly from the inlet towards the person should ideally be around 0.3 m/s and not deviate more than 0.5 m/s to ensure comfort. In addition, high velocities also result in excess noise. This is inappropriate for a cabin which should be as quiet as possible.

5.3.1 Idea 1

The idea starting with the exploration of having an air concept that allows for a top down approach. The first iteration was a basic one. The inlet is positioned above the person. The idea was to allow the air to flow downwards.

For this idea, there were 2 inlets, one inside the other. The inlet near the head has a velocity of 0.4 m/s and the series of inlet near the head has a velocity of about 0.9 m/s. Based on the area of the inlet holes, the total mass flow rate achieved was about 0.1 kg/s as stated earlier.

The inlet design looks like this, the inner circle is the airflow at lower velocity and the outer inlet is of higher velocity:

Figure 5-3: Inlet design for Idea 1

The reason for such a design is to vary the speed at which the air reaches the person. As the head is closer, the velocity from the inlet closer to the head does not need to be high. Likewise, as the body is further from the inlets, the velocity would need to be higher. Moreover, based on the research done by Zhang, in a hot environment, a cooled chest region would be optimal for comfort. Such an inlet system will allow for the person to adjust their inlet vents to control the speed and maybe temperature. So a total of 4 inlet vents will be positioned in the car for 4 persons.

Based on the preliminary simulation, the airflow pattern via velocity vectors was shown to be as such:

Figure 5-4: Idea 1 velocity vector plot

From the above figure, what is immediately noticeable is how differently the front seat and back seat differ in the air flow. The front seat air flow is being “pushed” inwards toward the person. This is due to the buoyant hot air from the hot dashboard moving upwards and flowing against the windscreen and thus affecting the air flow from the inlets.

Another interesting phenomenon can be seen at the air inlets for the one just above the head of the front passenger.

Figure 5-5: Idea 1 velocity vector plot close up

What can be seen is how the air from the back is flowing into the breathing region of the person and thus affecting the air flow from the inlet above the head. This means that the design of the inlets were not able to prevent the cross flow between the breathing space of

the person and the air outside this region. More interestingly, the higher inlet flow seemed to have drawn the air from the lower velocity inlet flow towards it. This meant that the air from the slow velocity inlet (the inlet above the head of the person) is not able to project air vertically downwards to the person. This means that the purpose of the slower inlet velocity is not achieved and is rendered quite useless.

This meant that the ideal 1 had major shortcomings which are not easy to solve. Therefore other ideas had to be explored. Two inlet vents having two different velocities would be not ideal as the flow of the low velocity air will be affected by the high velocity air due to the difference in pressure. This results in the inlet being unable to give a strong, reliable and directed flow towards the direction required.

5.3.2 Idea 2

Idea 2 consists of a slightly different concept. 2 types of inlets are found on the roof of the vehicle, one is directed at the airspace of the person, and the other is directed at cooling the entire space. The purpose of this is to see if it is more viable to cool in this manner and yet have a small PV portion to directly cool the person. The centre portion above the head would be the outlet.

The boundary conditions for the inlet are:

	Magnitude
Inlet (PV) temperature	292 K
Inlet (PV) velocity	3 m/s
Inlet (cabin) temperature	288 K
Inlet (cabin) velocity	3 m/s
Outlet	Pressure outlet

The method for driving the flow would be buoyant air flow. This would mean after the air flows from the inlet to the person, the air is heated by the hotter body and moves upwards due to the lower density. The air is supposed to move upwards towards the outlet vent.

Cooling the surrounding air is vital to allow for buoyancy to take effect. Otherwise, the air might not flow according to the ideal airflow path which is to be achieved. This is the reason for having 2 inlets.

The inlets on the roof will look something like the figure below:

Figure 5-6: Inlet design for idea 2

The inlet vents would be the rectangular slot and the small circular hole on the right. The rectangular vent would be used to cool the surrounding air, and the smaller circular hole will provide the targeted cooling towards the person. The outlet vent would be the larger circular outlet above the person.

Figure 5-7: Idea 2 velocity vector plot

From the velocity vector diagram above, a few issues can be seen. Firstly similar to the idea 1, the front passenger air flow is being “pushed” towards the person’s face. This means that the air flow is very much affected by the heat loading of the cabin, and it very easily affected by external factors. In fact, the movement of the buoyant air had also affected the second inlet (slot inlet), also pushing it towards the back of the vehicle.

Secondly, some of the air directly from the inlet to the outlet as the 2 is fairly close to each other resulting in a short cut of air flow. This can be considered to be a source of energy wastage.

One of the advantages in this case, is that cross contamination is minimised. One can see from the vector air flow, the air flow from one breathing space does not directly mix with the other breathing space of another passenger.

However one issue is that quite a substantial amount of mass flow rate into the space is needed to maintain the cabin at a cool temperature for the buoyant air to rise into the space. However, the flow of air towards the outlet is still more pressure based than by temperature convection. This can be noticed from the temperature distribution colour plot. In this colour plot, the general cabin temperature is about 24°C. This is in the comfortable range for the cabin comfort. Humidity was not taken into consideration in this simulation though.

5.3.3 Idea 3

Idea 3 was the initial progression to the final idea. For this idea, a simple inlet was placed that the top of the passenger's head in the roof of the cabin. This is to allow the air to flow downwards onto the passenger. From idea 1, after the realisation that the having differing velocities of air to allow for a more consistent flow reaching the passenger would not work as the lower velocity air would be deviated by the higher velocity air, especially if the inlets are very closed to each other.

As the objective is to attempt having a very low velocity flow rate yet having a normal mass flow rate of 0.1 kg/s for 4 persons, the inlet area would be larger. The temperature of the air coming into the space was tested at 3 temperatures: 5°C, 16°C and 18°C.

The initial test was air being pushed into the space at 5°C with a mass flow rate of about 0.06 kg/s. The airflow pattern was quite ideal shown to be able to maintain the airflow across the body of the person down to the abdomen. The inlet velocity is about 0.3 m/s.

Figure 5-8: Idea 3 temperature plot @ 5°C, 0.06 kg/s

Figure 5-9: Idea 3 Velocity vector plot @ 5°C, 0.06 kg/s

The main issue with this design is that although, the flow rate and temperature is low, having air of such cold temperature of 5°C would not be comfortable or seem appropriate for the air conditioning system design. Moreover, although the areas where the air was reaching was at 5°C, in some areas the temperatures exposed to the body were about 27°C. This is quite a huge temperature deviation in terms of temperature the body is exposed to. Therefore, the design had to cater to allowing for higher temperatures and if possible, lower temperature differences of the air the body might be exposed to.

One of the methods explored was to look at cooling the surrounding air and use a higher temperature at inlet. As such, the following method was looked at, this time the inlet temperature was at 18°C. The mass flow rate was more than doubled at 0.2 kg/s (twice the ideal mass flow rate of 0.1 kg/s). The velocity of the airflow from the inlet at the top of the person is about 0.25 m/s and the inlet for cooling the surrounding air is about 3 m/s.

Figure 5-10: Idea 3 temperature plot @ 18°C, 0.2 kg/s

Figure 5-11: Idea 3 velocity plot @ 18°C, 0.2 kg/s

Unlike the previous idea 2, the high velocity air inlet for cooling the cabin space is not affected much by the buoyant air from the dashboard. However it is clearly seen that the low velocity air from for the PV cooling of the front person is pushed back toward the person's face region. The deviation can be clearly seen when it is compared to the second person seated behind the person. Therefore, although the surrounding air is now cooled to a more comfortable 25°C, the PV inlet airflow is still very much affected by the warmer buoyant air from the dashboard. A method would be needed to minimise such effects.

One more variation was tested to see how the airflow responses to a slightly different air inlet temperature. The inlet temperature was lowered to 16°C. The velocity of the inlet above the person was 0.25 m/s, while the inlet to cool the cabin was at 2.5 m/s. Similar to the earlier idea, there is not much difference in the performance of the airflow. The airflow

pattern especially for the front person is still very much affected by the buoyant air even though the temperature of the inlet was reduced. Therefore a new method of delivering air to the person needs to be found.

Figure 5-12: Idea 3 temperature plot@ 16°C, 0.2 kg/s

Figure 5-13: Idea 3 velocity vector plot@ 16°C, 0.2 kg/s

5.3.4 Final design

Essentially, the most important property of the new design would be the ability to deliver air in a new way but with minimal effect from the warmer buoyant air from the dashboard. Most importantly, the air delivered should be too cold and/or too strong thus affecting comfort. If the airflow pattern is not affected, this would greatly allow for a well-controlled form of PV to the person or passengers in the vehicle.

This would be the goal of the design. This is accomplished by the development of a PV vehicle seat.

The design was created to ensure the PV airflow pattern was maintained and would not be affected by buoyant warmer airflows. In fact, more importantly, even at warm temperatures, the PV system should still be able to maintain a low flow rate with a relatively low velocity.

Figure 5-14: PV seat design

The reason for such a design is as follows, the seat has an arched back structure behind the person's head. This structure will restrict the inlet airflow movements maintaining the pressure of the person's airspace. This will prevent the airflow pattern from deviating and end up being pushed backwards, away from the desired path. In addition, with the air inlets moved a short distance below the ceiling. This allows the warmer buoyant air from the vehicle dashboard to move up along the ceiling of the vehicle. This means that the buoyant air will not interfere with the air coming in through the inlet leading to issues seen in the earlier designs. Thus by moving the inlet away from the ceiling but still above the person's head, the 2 issues major are solved.

The 2 curves at the side of the seat act as channels to direct the flow away towards the outlets underneath. In future designs however, the channel design can be optimized. The curve of the arch over the head is designed to bring the inlet on top of person. However the overhead length should not be excessive and become an obstruction of the view on the windscreen.

The seat is designed for a fully grown adult and in a relatively large vehicle. Therefore for the current design, it would be difficult to put this into a smaller vehicle. In addition, there should not be any issue with the seat accommodating different sized persons as the airflow is not greatly affected by the profile of the person.

Below is a preliminary result from the initial simulation:

Figure 5-15: PV seat design temperature and streamline plot

Note the high heat loads on the walls of the vehicle. However, the inlet air from the seat was not affected. This is a marked improvement from the previous designs. In addition, the inlet temperature was relatively high at 18°C. This would mean that this design would be able to work regardless of the temperature difference between the vehicle ambient air and the inlet air stream. The air flow is able to remain straight flowing downwards.

The most important issue at hand would be studying how this new design of ventilation performs compared to the conventional vehicle air conditioning. The comfort of the person and energy efficiency along with the airflow pattern will be used as a benchmark in order to assess the benefits and disadvantages of using such a design.

5.4 Simulated cases for PV in vehicle

The following cases are used to establish the performance of the PV system in a vehicle compared to the conventional air conditioning system. For this study, the aim was to achieve the same flow rate of air per person at a certain respective temperature. The 2 main criteria would be the thermal comfort of the manikin and the energy consumption of the system.

The use of the dedicated comfort manikin is to allow for a more accurate study of how comfort may be achieved with a PV system. A PV system has the potential to perform targeted cooling and setting up of a microclimate which is personalised for the occupant. This would allow for greater comfort, greater energy savings by cooling just the regions with occupants, and the ability to minimise the spread of contaminations.

For this iteration of the simulations, the seat which is the most vital component for this PV design will be mounted in a vehicle for the driver only. Solar radiation mimicking the sun will be shone on the vehicle. As the vehicle highest heat load will be due to the afternoon sun, the solar loading will not only be shining on exposed surfaces of the car like seats and the roof, but on the occupant's body also.

5.4.1 Case setup and boundary conditions

Based on the mesh setting and physics models studied during calibration, these settings were applied to the following PV simulations.

The vehicle layout is presented below:

Figure 5-16: Vehicle cabin space with PV seat

Figure 5-17: Convective cabin space

This vehicle is actually a simplified and smaller version of the earlier modelled Mercedes Vito. The vehicle has a similar volume as a usual big car. The conventional air conditioning is modelled using air conditioning performances and values from Jonsson, 2007. This is based on a Volvo. Although the vehicle shape might be slightly different, the volume and layout of the vehicles are similar.

Figure 5-18: PV seat placement with respect to the cabin

Figure 5-19: Convective seats placement with respect to cabin

The PV seat design is shown below:

Figure 5-20: PV seat front and side view

In fact, the layout and volume of the 2 models used for PV and the convention air conditioning are identical. This will provide for a fair comparison between the 2 systems.

The common boundary conditions for all the cases are as follows:

	Magnitude
Inlet temperature	292 K (20°C)
Direct solar intensity	1000W
Diffusive solar intensity	80W
Vehicle	
<ul style="list-style-type: none"> • Windscreen • Windows • Windscreen sidewall • Roof • Walls • Dashboard • Foot area • Floor • Back seat • Front passenger seat • Driver's seat 	<p>Greenglass-G90, PVB, Greenglass Greenglass</p> <p>Metal sheet, Fleece Board, PU-Foam (high density), PVC</p> <p>Metal sheet, PU-foam (low density II), Air, Fleece+CFK, PU-foam (low density II), Fleece Board+CFK, Décor (low density)</p> <p>1 Metal sheet, Bitum, PVC+PE-Foam, Air, Wood wool</p> <p>4 PU-foam high density, PU-foam medium density, PVC (low density)</p> <p>PU-foam (low density), Fire protection, Metal sheet, PU-foam (low density), PU+EPDM</p> <p>PVC (low density), Metal sheet, PU-foam (low density II), PU + EPDM, Carpet</p> <p>PU-foam (medium density), Fabric</p> <p>PU-foam (medium density), Fabric</p> <p>PU-foam (medium density), Fabric</p>
Inlet mass flow rate	0.02277kg/s (Average 0.34m/s)
Inlet temperature	Vary depending on case
Outlets	Pressure outlet
Manikin	Summer clothed

As stated at the start of this chapter, the simulation focuses on the comfort of the manikin and the energy usage of such a potential system. This allows for a common baseline which will allow for this PV system to be benchmarked against the conventional air-conditioning system.

For this simulation, only one manikin will be used due to the limitation in computation resources. However, this allows for a very good prediction of how such a PV system might perform and the type of changes which can improve on the existing design.

5.4.2 Physical models used in StarCCM simulations

The following physical models were selected for the StarCCM simulations. The mathematical models are explained in section 3.2.

Models	Details
Turbulence	$K - \varepsilon$
Buoyancy	On
Gravity	On (for buoyancy)
Solver	Segregated flow energy and momentum
Radiation Off	Off
Gas	Ideal gas
Surfaces	Non-slip
Gas type	Incompressible
Wall function	For all y^+
Multi-component gas	H_2O and Air (non reacting)
Simulation type	Implicit unsteady

5.4.3 Case 1: Conventional Air Conditioning at Medium Fan Speed

The conventional air conditioning specifications for this simulation was obtained from Jonsson. This case serves as the demonstration case of the standard air conditioning. This will assess the performance of the conventional air conditioning system and the type of comfort it can bring. The air patterns will also be studied to see the extent of the cross contamination between seats.

The conventional air conditioning consists of 4 inlet vents at the front of the vehicle. The air will blow inwards into the cabin. No pillar inlets were considered as usually pillar inlet will be used only as a supplement to the main inlets at the front of the vehicle.

Under heavy heat loads, the air conditioning is usually operated at a high flow rate with slower temperature. The velocity of the air was almost 0.9 m/s in some cases. In addition for this simulated case, the temperature was of about 8°C .

The boundary conditions for this case are as follows:

Parts	Magnitude
Velocity inlets Common boundary conditions <ul style="list-style-type: none"> • Extreme left dashboard • Extreme right dashboard • Middle left dashboard • Middle right dashboard 	Temperature: 8°C Mass fraction: $\frac{H_2O}{Air} = 0.00077$ Turbulence intensity: 0.01 Turbulence Viscosity Ratio: 10 Velocity: 1.64 m/s Inlet direction: (0.8644, 0.2994, 0.4039) Inlet direction: (0.8644, -0.2994, 0.4039) Inlet direction: (0.847, -0.352, 0.3979) Inlet direction: (0.8424, 0.3014, 0.4067)
Pressure outlet	0 Pa
All other surfaces	Temperature boundary conditions (depending on Theseus FE coupling)

5.4.4 Case 2: Conventional Air Conditioning at Medium Fan Speed with air conditioning directed at only one person.

The layout of the vehicle is the same as case 1. However, in this case, the assumption is that only the inlets nearest to the person is switched on allowing for airflow into the cabin. The mass airflow rate is targeted at no more than 0.025 kg/s as only 1 person is to be cooled. This simulation will serve as a benchmark into to understand if the conventional airflow allow for the person to be more comfortable under the existing solar loading.

The 2 airflow at the front will have high velocities compared to the typical PV systems. Using the similar values from the reference simulation, a temperature of 8°C was used.

With that changed, all the rest of the simulation boundary conditions remain.

Parts	Magnitude
Velocity inlets Common boundary conditions <ul style="list-style-type: none"> • Extreme right dashboard • Middle right dashboard 	Temperature: 8°C Mass fraction: $\frac{H_2O}{Air} = 0.00077$ Turbulence intensity: 0.01 Turbulence Viscosity Ratio: 10 Velocity: 1.64 m/s Inlet direction: (0.8644, -0.2994, 0.4039) Inlet direction: (0.8424, 0.3014, 0.4067)
Pressure outlet	0 Pa
All other surfaces	Temperature boundary conditions (depending on Theseus FE coupling)

The body surface was also made as a N_2 source producing $0.002 \text{ kg/m}^2\text{s}$ of N_2 . This will assist to study in the potential contaminant flow.

5.4.5 Case 3: PV Seat Air Conditioning at Medium Fan Speed for only one person.

The layout of the vehicle is the same as the previous cases allowing for 4 persons to be seated. The only difference is that the driver seat is now of the PV type. This means that the inlets are situated on the seat itself as per the design concept.

For the PV based designed cooling concepts, the aim is to have a cooling system that does not require high velocity or low temperatures. This will serve to improve comfort and yet provide a lower energy solution essential for electric vehicles.

The design objective is to only cool the zones where people are present to control the temperature in those zones. The PV concept will be tested to understand its ability in providing a micro-climate which allows the people to enjoy their place with delicate temperature control.

The velocity is at 0.04 m/s which is stated to be the max velocity which the person exposed to such air, would be unable to feel the air movement. The temperature is also higher at 18°C . This ensures that the person does not have cold air blown harshly toward him or his face. The mass flow rate is maintained at a constant value of no more than 0.025 kg/s in order to achieve a total flow rate of 0.1 kg/s for 4 persons.

The boundary conditions are as follows:

Parts	Magnitude
Mass flow inlets	Temperature: 18°C Mass fraction: $\frac{H_2O}{Air} = 0.00077$ Turbulence intensity: 0.01 Turbulence Viscosity Ratio: 10 Inlet direction: Normal
<ul style="list-style-type: none"> Left seat inlet 	Mass flow: 0.00715 kg/s (0.03 m/s)
<ul style="list-style-type: none"> Right seat inlet 	Mass flow: 0.00715 kg/s (0.03 m/s)
<ul style="list-style-type: none"> Middle seat inlet 	Mass flow: 0.00847 kg/s (0.03 m/s)
Pressure outlet	0 Pa
All other surfaces	Temperature boundary conditions (depending on Theseus FE coupling)

Similar to case 2, the body surface was also made as a N_2 source producing $0.002 \text{ kg/m}^2\text{s}$ of N_2 . This will assist to study in the potential contaminant flow.

5.5 Results

The results from the 3 cases will be presented here in sequential order. The total computation time for each of the cases was approximately 3 week for each case.

The long computation time was due to the coupling process between the 2 simulation programmes, StarCCM and Theseus FE. The transfer of data took about half of the total simulation time, while the other half was taken up by the calculation of the simulation variables. The simulation was run on two 4 core I7 Intel Xeon CPU with 32 GB of ram. This computer was one of the more powerful personal workstation for simulation in 2012.

The results will look at the following important parameters:

1. Surface temperatures in the vehicle
2. Airflow patterns of the air
3. Air Temperature in the vehicle and of the manikin
4. Overall comfort parameters
5. H_2O distribution in the vehicle

For all the cases, the residuals for the momentum, continuity, turbulent kinetic energy and H_2O mass fraction were below 0.001. The energy equation was at the order of 0.01 and the turbulence dissipation rate was at the order of 0.1. Due to the nature of a co-simulation simulation, the timestep were not made smaller then 1.0s.

For the simulation, the temperature at various points was monitored until it reached steady state. The simulation time for all the cases was set at 3600s or 1hour.

5.5.1 Case 1: Conventional Air Conditioning at Medium Fan Speed

This case presents the basic vehicle air condition performance of a typical vehicle. This vehicle air has 4 inlets blowing inwards to the cabin. The streamlines into the cabin blows straight into the central area of the airspace, essentially providing the airspace with moving air.

Figure 5-21: Streamline with temperature for conventional air conditioning with all inlet operating

As the air leaves the inlets and moves into the airzone of the cabin, the air gets warmer. Starting from 8°C, the air enters the outlet at about 28°C. This shows how warm the airspace is resulting in the transfer of heat to the airspace. Most importantly, the streamlines shows that the air will flow towards the front seats cooling the frontal area and then move into the back part of the cabin airzone. Thus air from the inlet crosses from the front region to the back region and moves under the seats and back to the outlets at the front under the dashboards.

Figure 5-22: Manikin surface temperatures for conventional air conditioning layout

The temperatures on the manikin range from about 30°C to 36°C. The frontal region of the manikin has relatively higher temperatures compared to the limbs. The head is also one of the areas with higher temperatures.

Figure 5-23 presents the temperatures of the surfaces in the vehicle. Note that even with the air conditioning system on and active, the interior surfaces of the vehicle can still reach very high temperatures. Temperature of surfaces exposed to solar radiation like the dashboard and the front seats reached temperatures of about 50°C to 65°C. The temperatures without no radiation exposure were much lower, generally below 40°C.

Figure 5-23: Temperature of surfaces in cabin for conventional air conditioning layout

The temperatures range from about 50°C to 60°C for windows exposed to the sun. Windows on the right of the vehicle and the windscreen will be of higher temperature due to the higher radiation exposure due to the sun's position.

Figure 5-24: External surfaces temperature for conventional air conditioning layout

The air temperatures in the cabin around the manikin are quite low due to the air inlet temperatures blowing in. Notice similar to the preliminary simulations, the air around the surface of the windscreen is very warm at about 38°C and as it moves upwards to the ceiling it goes to about 40°C. Therefore a huge heat load comes from the high temperatures near the front dashboard and glass surface and ceiling.

However note how the temperatures near the cabin area direct in line of the air inlet area are about 17 – 24°C. This is shown for 2 of the planes sliced parallel to the vehicle front and sides respectively. An example of the plane slices with respect to the model is shown in Appendix D. Note that the warmer temperatures tend to be near to the ceiling of the cabin and the cooler air is in the lower regions of the cabin.

Figure 5-25: Temperature side contour plot for conventional air conditioning layout

Figure 5-26: Temperature front contour plot for conventional air conditioning layout

With the vectors, the air flow path can be seen. 2 main areas to take note would be the rising of air at the front of the dashboard and the airflow from the inlet towards the centre of the cabin. There is also recirculation at the front of the manikin. The velocity of the air leaving the inlet is about $1.8 - 2.0 \text{ m/s}$. When it reaches the back of the cabin, the velocity drops to about 0.8 m/s .

Figure 5-27: Streamlines with velocity for convective air conditioning layout

Figure 5-28: Velocity vector side plot for conventional air conditioning layout

Figure 5-29: Velocity vector front plot for conventional air conditioning layout

The humidity content of the air space gives a gauge of how contaminations might flow. The humidity from the manikin can be seen moving from the manikin towards the back area of the cabin. The moisture can be seen being produced by the manikin. However the moisture content being produced by the manikin is extremely low.

Figure 5-30: Mass fraction plot for H_2O convectonal air conditioning layout

Figure 5-31: Mass fraction side plot for H_2O convectonal air conditioning layout

The figure below shows the sensation perceived by the manikin using data from Zhang's research. Below is the scale used for Zhang's sensation and comfort indices respectively.

Figure 5-32: Zhang's comfort and sensation indices (extracted from Theseus FE theory manual)

The manikin basically found that the airzone temperature and humidity quite comfortable although on the warm side for most of the body parts. Most value for sensation range above 0 except of the head and the left arm which was directly in the air inlet path resulting in feeling cold.

Figure 5-33: Zhang's Sensation Index for conventional air conditioning layout

Likewise, for the comfort, most of the parts are in the comfortable region as the values of the limbs are around zero. However for the chest and abdomen area, the comfort was perceived to be uncomfortable with a value of about 2.

Figure 5-34: Zhang's Comfort Index for conventional air conditioning layout

As for the equivalent temperature, the manikin temperatures reach above 30°C for many parts of the body especially the chest area. These temperatures are considered high. The effects can be seen for the ISO 14505-2 results for comfort.

Figure 5-35: Local equivalent temperature for conventional air conditioning layout

For the local thermal comfort zone based on ISO 14505-2, the body perceived the temperature as uncomfortable and being too warm reaching the highest value of 5 for many parts of the body.

Figure 5-36: ISO 14505-2 comfort index for conventional air conditioning layout

By combining the local Zhang’s sensation results to obtain the overall manikin sensation reading, the sensation index was found to be -0.6 . This meant that the manikin felt that it was cool despite the heavy solar radiation loading in the vehicle.

Figure 5-37: Overall Zhang sensation index for conventional air conditioning layout

The overall Zhang comfort index gives a value of -0.08. This is very close to the thermal comfort equivalent of just comfortable.

Figure 5-38: Overall Zhang comfort index for conventional air conditioning layout

Moreover, the constant which controls sweating, S_w has a value of 0 from the start to the end of the simulation. This means that the manikin did not feel hot and there was no need for the body to remove excess heat from the body via sweating. This would be the ideal case for the manikin.

5.5.2 Case 2: PV seat with inlet velocity of 0.4 m/s with temperature of 8°C

A case with an inlet velocity for the PV system was run at 0.3 m/s with a temperature of 8°C. However the final condition of the manikin was found to be slightly worse than this case with poorer comfort values.

Therefore only the case with 0.4 m/s with 8°C will be discussed in this report.

From the figure below, it can be seen that the streamlines from PV inlets at 0.4 m/s is able to produce an ideal airflow that flows along the contour of the person. There is relatively low flow in the other regions of the cabin space. The temperatures of the air stream start of about 8°C and progresses to higher temperatures after leaving the body.

Note that this is an extreme case of cooling with only the single seat cooling functioning. The purpose of this simulation is to compare the convectioal air condition setup with the PV type. Using the same flow rate and temperature the simulations will show if this seat can improve the temperatures and cooling for the vehicle.

Figure 5-39: Streamline with temperature for single PV inlet operating

Below the temperatures of manikin surface can be seen. Notice that the manikin surface temperatures can reach temperatures of about 40°C. The lowest temperature was around the head region where the inlets were of about 34°C.

Figure 5-40: Manikin surface temperatures for single PV inlet operating

The temperature of the cabin surfaces show that despite the harsh heat loading, the cooler surface temperatures were found around the PV seat region. The cooler temperatures were about 34 – 42°C compared to some areas reaching a temperature of over 60°C.

Figure 5-41: Temperature of surfaces in cabin for single PV inlet operating

Figure 5-42: External surfaces temperature for cabin for single PV inlet operating

In this temperature colour contour with reference to Figure 5-43, the temperature regions can be seen clearly. Around the body region, the temperature was about 8 – 20°C. This means the temperature zone around the body was relatively low compared to the surrounding region.

Figure 5-43: Temperature side contour plot for single PV inlet operating

Similarly, directly near the body of the manikin, the temperature is quite low compared to the other regions, ranging from 25 – 30°C.

Figure 5-44: Temperature front contour plot for single PV inlet operating

It is easily noticeable that the streamline for velocity and temperature is very similar. The velocity changed quite similarly only having a gradual change from the inlet to the cabin area, going from 0.4 m/s to about 0.79 m/s , and getting very close to zero as the air loses momentum.

Figure 5-45: Streamlines with velocity for single PV inlet operating

Similarly, the motion of the air particles can be seen moving from the inlet to the body and downwards. What is most important to take note is that, the airstream is not very much affected by the heat load from the surface at the dashboard. Speeds along the body increased to about 0.8 m/s as the air flow reached further away from the body.

Figure 5-46: Velocity vector side plot for single PV inlet operating

Figure 5-47: Velocity vector front plot for single PV inlet operating

The mass fraction of H_2O is very much controlled within the local airspace of the manikin. The moisture around the manikin shows how the parts are producing moisture in response to the environment. The mass fraction is about 0.005 for most parts of the cabin.

Figure 5-48: Mass fraction plot for H_2O for single PV inlet operating

Figure 5-49: Mass fraction side plot for H_2O for single PV inlet operating

Looking at the N_2 mass fraction of the same plane focusing on the back seat region, it can be still that the breathing zone for the passengers at the back seat is exposed to very little N_2 produced by the manikin at the front. The area at the back seat had a mass fraction of about 0.0718.

Figure 5-50: Mass fraction of N_2 for side plane slice for single PV inlet operating

The plot for Zhang's sensation shows that the chest and abdomen areas of the manikin is feeling between neutral and warm with sensation values ranging from 1 to 2 (the sensation index range from -4 being very cold to 0 for neutral to 4 being very hot). The arms are quite on the hot side having values more than 3. The head is feeling cool having a value of about -1.

Figure 5-51: Zhang's Sensation Index for single PV inlet operating

In terms of local comfort, most of the chest and abdomen areas would be considered feeling just comfortable to slightly uncomfortable. This is especially the areas that are in contact with the solar radiation. The head would be feeling comfortable. The arms would be one of the few parts feeling quite uncomfortable with a value above 3.

Figure 5-52: Zhang's Comfort Index for single PV inlet operating

If local equivalent temperature is used, the temperatures calculated are quite high. In fact, most body parts are in the range of 40 – 50°C which is quite high. This brings about a score of 5 (too warm, uncomfortable) for comfort using the ISO 14505-2 standard.

Figure 5-53: Local equivalent temperature for single PV inlet operating

Figure 5-54: ISO 14505-2 comfort index for single PV inlet operating

The overall Zhang's comfort index was obtained from the local values. This gave a value of about -1.34 . This put the manikin in a region of slightly uncomfortable.

Figure 5-55: Overall Zhang Comfort for single PV inlet operating

The comfort value of -1.34 can be explained by the high sensation index of a value 1.6 . This puts the manikin in the region of feeling warm when seated in the cabin.

Figure 5-56: Overall Zhang sensation for single PV inlet operating

From the SW constant which controls the sweating of the manikin, it can be seen that the manikin has been sweating slightly. This is the thermoregulation response which would be seen when the manikin is in an environment which feels hot.

Figure 5-57: Sw constant which controls the rate of sweating for single PV inlet operating

5.5.3 Case 3: PV seat with inlet velocity of 0.84 m/s with temperature of 8°C

In order to test the effectiveness of the PV type of cooling system for the vehicle, it was benchmarked against the conventional air-conditioning system. However, in order to ensure only the method of airflow was tested, the same mass flow rate and temperature of air was allowing into the cabin.

For this simulation, the inlet had a flow rate with a velocity of 0.84 m/s. This means that the airflow for the convention system would have higher inertia compared to the PV system which is flowing with lower velocity.

The inlets to the cabin were the 2 nearest vents to the manikin. This would allow for direct flow towards the manikin. Most important, this will show if there would be a difference maintaining temperatures and comfort values during cooling for such a configuration.

From the streamlines, the convention system was able to blow air directly towards the front of the person with a high velocity of 0.84 m/s. The velocity drops to about 0.3 m/s after impact with the manikin and flows around the cabin. Note that despite the high velocity, the inlet air curves downwards at about 45° towards the manikin.

Figure 5-58: Streamline with temperature for conventional air conditioning with 2 inlets operating

Interestingly, only the upper limb has a very high temperature of about 50°C. The other parts of the body were cooler with the chest having a slightly higher value of about 46°C.

Figure 5-59: Manikin surface temperatures for conventional air conditioning with 2 inlets operating

Looking the overall surface temperatures, the dashboard is able to reach a very high temperature of about 81°C. The other parts of the vehicle had also high temperatures above 70°C, especially the parts affected by radiation.

Figure 5-60: Temperature of surfaces in cabin for conventional air conditioning with 2 inlets operating

From the color plot, it is shown that the air impacting the manikin is at about 15 – 20°C. However, the jet impacts at the upper abdomen area. It can be seen from the previous streamline plot, the airflow bends downwards, as it leaves the inlet.

Figure 5-61: External surfaces temperature for cabin for conventional air conditioning with 2 inlets operating

Figure 5-62: Temperature side contour plot for conventional air conditioning with 2 inlets operating

Likewise from the below contour plot, the temperature of the air around the manikin is about 20 – 25°C.

Figure 5-63: Temperature front contour plot for conventional air conditioning with 2 inlets operating

The streamline plot below shows that when the air stream hits the manikin, the temperature increases to about 30 – 40°C.

Figure 5-64: Streamlines with velocity for conventional air conditioning with 2 inlets operating

The velocity vector plot shows 2 characteristics of the air flow. Firstly, there is a thermal plume above the manikin. This is due to the plume leaving the head region due to sweating of the manikin. This will be explained in more detail in the mass fraction plot.

Secondly would be how the air travels from the inlet to the outlet. In this setup, the air from the inlet did not go in a direct path. In fact for this case as mentioned earlier, the air movement was in a slight parabolic curve downwards.

Figure 5-65: Velocity vector side plot for conventional air conditioning with 2 inlets operating

Figure 5-66: Velocity vector front plot for conventional air conditioning with 2 inlets operating

As for the mass fraction, it can be seen how the moisture moves with respect to the airflow. What is immediately noticeable is that a high proportion of moisture goes upwards towards the ceiling of the cabin. The mass fraction of H_2O is quite high exceeding 0.02. The moisture can also be found moving throughout the cabin space.

Figure 5-67: Mass fraction plot for H_2O for conventional air conditioning with 2 inlets operating

Figure 5-68: Mass fraction side plot for H_2O for conventional air conditioning with 2 inlets operating

The N_2 contamination study showing the spread of contaminants in the cabin showed that the N_2 moved from the front of the cabin to the back seat region. The N_2 mass fraction at the back seat breathing zone height was about 0.0955

Figure 5-69: Mass fraction of N_2 for side plane slice of conventional air conditioning layout

In terms of the local sensation, the head region feels too warm with a sensation index of 3 – 4. In fact the upper regions of the body have index stating that it is on the warm side. The lower part of the body is only slightly warm with index values ranging from 0 – 1.5.

As for the comfort aspect, most of the head area was feeling uncomfortable, in fact feeling very uncomfortable. The chest area is considered uncomfortable too. However the lower half of the body was considered comfortable with an index below 0. These results are expected judging from the sensation index obtained.

Figure 5-70: Zhang's Sensation Index for conventional air conditioning with 2 inlets operating

Figure 5-71: Zhang's Comfort Index for conventional air conditioning with 2 inlets operating

The equivalent plot gives shows that the temperature is quite high with the head area reaching temperatures above 65°C . The other regions are also quite high for the upper body, ranging from $50 - 60^{\circ}\text{C}$. Likewise for the lower portion of the body, the temperatures are much lower.

Figure 5-72: Local equivalent temperature for conventional air conditioning with 2 inlets operating

The ISO 14505-2 comfort level shows that the manikin would regard the environment as too warm, having an index value of 5.

Figure 5-73: ISO 14505-2 comfort index for conventional air conditioning with 2 inlets operating

The overall Zhang's sensation index is about 1.5 – 2.4 being in the very warm region. It was possible to see a fluctuation in the sensation region. This is attributed to the sweating mechanism stepping into regulate the sensation and comfort of the body.

Figure 5-74: Overall Zhang's sensation index for conventional air conditioning with 2 inlets operating

The comfort value fluctuates between –1.35 to –2, in tandem with the sensation index.

Figure 5-75: Overall Zhang's comfort index for conventional air conditioning with 2 inlets operating

From the SW constant which controls the sweating of the manikin, it can be seen that the manikin has been quite considerably, reaching a max value of 2.7. The sweating regulation occurs providing cooling, as the Sw constant changes, so does the body temperature and thus the comfort index level.

Figure 5-76: Sw constant which controls the rate of sweating for conventional air conditioning with 2 inlets operating

5.6 Discussion

The 3 simulated cases allowed for a better understanding of the performance of a PV system in use of a car, its limitations and advantages. The first simulation, with full air conditioning running with a single person heat load and solar radiation serves as a benchmark for comfort under such heat loading. The 2 other simulations, one is of the PV system itself and the other is of the conventional system. Due to the time limitations, only these few cases were ran.

The inlet conditions of the PV system and the conventional system are the same, except the velocity of the airflow. This means that the essential condition like that this allows for a fairer comparison between the 2 cases, differing only in the position of the inlets and the velocity, both are the characteristics for a PV type of system.

The hypothesis would be the use of a PV system should be able to enhance the comfort level and yet save on energy requirements if properly designed for a certain heat load when compared to conventional cooling methods. PV systems are meant to bring the airflow to where it is required to provide for an effective and efficient cooling method.

5.6.1 Comfort

5.6.1.1 Case 1 (benchmark case)

Even with the high solar loading of $1000W$ and larger than usual windows, the air conditioning inlet conditions were able to cool the environment to a comfortable average temperature of about $26.7^{\circ}C$. This is with the full mass flow rate of 0.1 kg/s and a temperature of $8^{\circ}C$.

The streamlines shows how the air will circulate in the cabin space, moving from the front region of the vehicle to the back. The air would be considered moving from region to region with no restrictions. This allows for a relatively uniform temperature on all the surfaces. In fact due to the high flow rate and low temperatures, this allowed a large heat load to be removed from the space.

Figure 5-77: Temperature side contour plot for conventional air conditioning layout

Noticed the air zone region which the person is situated in has a temperature of only 24°C. Air at this temperature compasses the entire manikin. This means that the entire manikin is totally cooled to a comfortable temperature even under the high solar radiation load. However note that this mass flow rate for this simulation is only meant one person. Due to the position and location of the inlet vents, it is not possible to have any form of control in the airflow pattern. It would have been ideal if it was possible to focus all the cooling mechanisms such as inlets towards the region where passengers are located. Therefore PV type of cooling benefits is hard to achieve as the airflow cannot be constrained.

The conventional air conditioning with such inlet conditions also shows that it is able to achieve very ideal surface temperatures even lower than the body's core temperature of 36.9°C. This would be the main reason which results in the comfort value obtained being close to zero signifying that the manikin felt comfortable.

Under such an extreme solar loading and radiation exposure of the cabin due to the large windows, attaining a comfort index of just comfortable is definitely a good result. However, definitely a lot more energy input would be required for the manikin to obtain a result of +4 which gives a comfort state of very comfortable. This would definitely be an index score every air conditioning system should aim to attain.

However with large windows and high solar loading compromises between comfort and energy usage would need to be balanced.

Figure 5-78: Thermal sensation and comfort readings for case 1

If the local sensations were looked into detail, it is noticeable that most body parts feels in a slightly warm having values above 0 to 1.76. However the extremities are on the cooler side have a negative value thus feeling cooler. This can be due to the extremities, such as the arms, are in the flow path of the air.

Likewise, this resulted in the comfort of the manikin having the chest and abdomen areas having a comfort rating of 2 signifying that the core body region is feeling comfortable. The arms are feeling quite uncomfortable and this would be attributed to the cold temperature

of the air conditioning air flow inlet positioning. Otherwise most regions of the body were found to be in a comfortable region.

The main advantages of the conventional air conditioning layout would definitely be the ability of the system to cool the entire cabin space. It brings very quick relieve to the air space temperature cooling all the surfaces in the vehicle thus minimising the re-radiation of the thermal radiation to the manikin.

The major disadvantage would be that this cooling strategy can be considered to have high energy usage with low inlet temperatures and high flow rates. Although it was capped at 0.1 kg/s , this method is considered will cool the entire airspace with little or no regard in focusing only on the areas with persons present. In other words, the air flow is difficult to regulate or control. It might be possible to just control the airflow by switching on or off some of the inlets depending on the presence of people. However, due to the nature of the airflow from these inlets, the air will just flow pass the person towards others or recirculate. This is evident when studying the streamlines of the airflow.

Figure 5-79: Streamlines plot for case 1

From the figure above, it is noticeable that the air flows towards the manikin driver, and passes over him to the back region of the cabin. This shows the lack of control of the air flow which is a characteristic for a horizontal type of airflow at high velocities.

This would be where a PV type of system with a top down type of airflow may be advantageous .

In this case, it is also possible to see the comfort results which were obtained by the 2 other comfort indexes, namely the ISO 14505-2 and the PPD. It was found that the 2 other indexes were much more stringent in terms of comfort.

For the ISO 14505-2 comfort index shows a very drastic difference in comfort. Even though the manikin attains a comfortable rating for Zhang's comfort index, it shown here that for the ISO 14505-2 index, it fares poorly. This can be attributed to the high solar radiation load affecting the local equivalent temperatures. Partly, most comfort indexes were not designed for high stratified temperature loading such as in a vehicle. Vehicle heat loadings are one of the more extreme heats loading a human would need to cope with and one that will require effective climatisation methods to cool the space.

Figure 5-80: ISO 14505-2 comfort index for case 1

Based on the PMV and PPD scores for case 1, it is shown that PMV was calculated to be 4.3 which is based on the 7 point ASHRAE scale. For this scale anything above +3 is considered hot, which implies that the environment is considered to be very warm.

Due to this, the PPD is calculated to be 100%. A PPD of 100% would mean a very high dissatisfied rate and a high degree of discomfort. From this results, it can be concluded that even for the conventional air-conditioning running on the full targeted mass flow rate of 0.1 kg/s with only one person at 8°C , using the “traditional” comfort scales, the comfort value is already exceeded the max value.

It is importantly to note that the PPD, PMV and the ISO 14505-2 were designed to be used for not so extreme temperature conditions. Moreover, in the test cases for this vehicle, the temperatures are stratified and some parts of the body do face a much higher heat load than others.

However, Zhang's comfort index was a result of many tests focusing on every individual part of the body and testing for the resulting comfort and sensation. It was designed with

temperature stratification in mind and thus it able to cope with more extreme temperature conditions.

To conclude for the analysis of case 2 and case 3, the PPD, PMV would not be discussed as these cases having even lower air flow into the cabin, can only result in even higher discomfort and dissatisfied results. These results would be meaningless when it exceeds the scales for these indexes.

5.6.1.2 Case 1 and case 3

In this section, Case 3 will be discussed and case 1 will be compared with case 3 to evaluate the results shown and what might be concluded.

Case 3 is similar to case 1 only that the inlet mass flow rate has been limited to the mass flow rate for 1 person. The rational for using a 1 person mass flow rate would be that in the ideal situation the mass flow rate required by the air conditioning might be considered to be of a linear relationship with the number of persons. In order words, if the total flow rate deliverable by the air conditioning is 0.1 kg/s. For one person, the mass flow rate should be 0.1 kg/s divided by 4, if the total number of people is regarded as 4. Thus if there are 2 persons, the mass flow rate will be 0.05 kg/s.

However, this might not be the case in reality. It might be more essential to the aspect of comfort, for the mass flow rate to go via a natural log relationship with the number of person. This can be better explained at the end of this section.

Nevertheless, by ensuring that the mass flow rates between the PV case and conventional case had the same mass flow rates and inlet temperature, would be a fair way to assess if the method of controlling and blowing of the air would result in differing results in terms of energy usage and comfort.

For case 3 and case 1 uses back the same cabin layout. As mention above, only the inlet conditions have been changed to provide the means to compare between the cooling system. With the change in inlet conditions, the results differed significantly with the decrease of mass flow rate.

Below, the change in the streamlines pattern is the most apparent. With the drop in mass flow rate limiting to only 1 person, the performance of the inlet drops drastically.

Figure 5-81: Streamline comparison between case 1 and case 3

As shown above figure above, the streamline for case 3 is only able to reach a throw distance that is half of the case 1. One way to overcome this issue might be to increase the velocity of the inlet. In order to do so without the increase mass flow rate would be by changing the inlet area. However, that would be another issue altogether.

Case 3 results in the air losing velocity in the horizontal direction; this is due to a combination of buoyancy effects of the warm environment and the lower inertia of the air due to the lower speed. The buoyancy effects of the warmer environment results in the warmer air rising and the denser cooler air sinking towards the ground.

Due to this effect, the surfaces of the manikin cooled are directly affected. This is profoundly seen on the surface temperatures of the manikin. Due to how the air curves downwards towards the manikin, the lower region of the manikin is cooled more significantly compared to the top half. This can be seen in the surface temperatures and then on the sensation index.

Figure 5-82: Temperature and comfort comparison between case 1 and case 3

The surfaces of case 1 significantly lower at about 4 – 9°C compared to case 3. This trend can be seen for the comfort index of each region too. Discomfort can only be found on the left arm and the underside of the right arm for case 1, as for case 3 the discomfort is more widespread. The head, the arms and the chest area all falls under the discomfort region. In fact, the head region has the highest discomfort index value of –4.

Judging from the temperature of the air from Figure 5-83, the temperatures at the upper portion of the cabin air is warmer in green. For case 1, the warmer temperatures are found to be 38 – 46°C. However of case 3, the temperatures are about 45 – 58 °C. However, what makes case 3 much worst in terms of high temperature air is that the amount of warm air which can be found in the cabin space. Based on the plane slice area alone, the amount of warmer air for case 3 is about 50% more than case 1, and it is of a higher temperature.

Figure 5-83: Temperature from side plane of case 1 and case 3

Therefore the lack of mass flow rate into the airspace has a great adverse effect on the temperatures and thus the comfort level of the manikin.

5.6.2 Comparison between Case 2 and 3

Case 2 is a PV type of cooling making use of an integrated seat with inlets to bring cool air to manikin. The temperature and the mass flow rate into the cabin is the same as the conventional air conditioning case 3. The only difference is the velocity of the airflow and the location of the inlets. The inlets were optimised to allow a top down type of airflow. This has a few advantages compared to case 3 and will be discussed in this section.

Figure 5-84: Streamlines Case 2 seat inlets

The streamlines of case 2 shows that the PV seat inlets are able to sustain an uninterrupted flow from the inlet to the manikin body following the contour of the body. A high percentage of the air is focused on the manikin surface with little “leakage”. Most of the airflow moves downwards to the floor and circulates there, before going to the outlet to be extracted from the cabin space.

However, similar to case 3, the temperature of the streamlines increases quickly from the inlet to the manikin surface. The temperature changes from 8°C to about 23°C as it leaves the manikin surface due to the high temperatures in the airspace.

This can be clearly seen as the air leaves further from the manikin, the temperature increases above 40°C as it circulates around the cabin area.

Figure 5-85: Manikin surface temperatures case 2 vs case 3

Even the 2 air conditioning systems has the same mass flow rate, the surface temperatures attained by the manikin is quite different. Focusing on each major individual body parts, the difference between the 2 cases is quite significant. For most body parts, Case 3 has higher skin temperatures compared to case 2. The percentage increase between case 2 and case 3, range from 0% – 17%. One major body part case 3 has a lower temperature would be the abdomen. The percentage decrease in the temperature is about 7%.

The lower region of the manikin tends to have similar or slightly lower temperatures for case 3 compared to case 2. This is due to the direction of inlet air flow. The first position of impact of the airflow for case 3 would be abdomen region and below, while for the case 2, the first position of impact would be the head and upper region.

Figure 5-86: Air temperature of plane slice case 2 and case 3

Focusing on the temperature of the air around the manikin with reference to the plane slice, the area around the manikin is actually much cooler for case 2 compared to case 3. In fact, it can be seen for case 2, the temperatures around the chest and abdomen areas is about 20°C

compared to case 3 where the chest region having both cool air at 24°C and the warmer air around about 40°C. The zone around the head around for case 2 is about 30°C compared to case 3 with temperatures of above 45°C.

Figure 5-87: Frontal plane slice for air temperature case 2 and case 3

With comparing the frontal plane slice for case 2 and case 3, it can be seen that case 2 is able to “wrap” the manikin body with cool air of about 25°C. Case 3 is unable to do so and there is a lot of air flowing out of the seat in to the surroundings. However this plane slice presents the results for the lower portion of the body. It can be seen that the temperatures are quite cool at about 24°C.

Therefore from these results, it can be seen that the PV system, case 2, is able to cool the major body parts like the head, the chest and the abdomen more consistently. Case 3 is able to cool the abdominal region better as the inlet is focused at that region of the body. However for the chest and hand, there is no direct cooling.

The effectiveness of the cooling of these parts greatly affect the comfort and sensation felt by the manikin. As shown in the previous cases, there is a direct correlation between the temperatures of the air and the skin surfaces with the comfort of the manikin.

Similar to the temperatures of the skin surface, the local comfort based on the Zhang’s comfort index shows the similar regions with high discomfort. For case 3, the head had the highest discomfort value of about –4 compared to case 2 which were in the comfortable region. Likewise, the chest area also showed the same trend. For both case 3 and case 2, the abdomen area had similar results being in the comfortable region.

However for case 2, it can be seen that the arms has a very high discomfort index score. This can be attributed to the total lack of cooling to the arms. Although in terms of comfort, the arms can be allow reaching a higher temperature. However in this case, the lack of cooling resulted in the surface temperatures increasing a little too much. This would be a concern for future design variations of the seat.

Figure 5-88: Zhang's comfort index for case 2 and case 3

With regards to the overall sensation and comfort score, case 2 has a stable overall sensation score of about 1.6 compared to case 3 with a varying score between 1.5 – 2.4. This means that the manikin felt slightly warm for case 2 but warm in case 3.

As for comfort, the Zhang's comfort index gives a index score of about -1.34 compared to case 3, which has a value of -1.35 to -2 . The comfort for case 2 is also better than then case 3, as the higher the comfort index score, the more comfortable. This would be due to the significant areas with high discomfort for case 3 such as the head and the chest. For case 2, the score should improve significantly if the lower arms can be properly cooled and the surface temperature lowered.

Another issue would be that case 3 has resulted in the manikin sweating significantly in order to cool the body down as the hotter areas are in the vital parts of the body, such as the chest and head. This results in the comfort fluctuations. This means that the airspace temperature has reached too high values resulting in an unsustainable living condition.

Figure 5-89: Mass fraction of H_2O for case 2 and case 3

Figure 5-90: Mass fraction of H_2O for case 3 capped at 0.01

2 things are immediately noticeable when the mass fraction plot from the side plane observed. Firstly, the H_2O mass fraction is much higher for case 3 than case 2, confirming that the manikin is sweating much more for case 3 than case 2. This would be attributed to the higher temperature the manikin would need to endure. Secondly, the flow of the moisture around the cabin is limited compared to case 3. Case 3 moisture flows from the front region to the back. This is potentially the breathing space of back passengers. For case 2, there is little or no sign of moisture from the manikin reaching the back of the vehicle. This would signify that the cross contamination of the breathing spaces between passengers will be very much minimised compared to case 3.

Figure 5-91: Mass fraction of N_2 for case 2 and case 3

The same trend can be seen for the N_2 spread in the cabin. Similar to the H_2O particles movement, it can be seen that the N_2 mass fraction for case 2 is lower than case 3. The breathing zone point marked with a red dot in Figure 5-91 had an N_2 mass fraction of 0.0718 for case 2 and 0.0954 for case 3. This meant an improvement of 32% for case 2 showing that the air flow in case 2 (PV system) is better controlled. If there is a passenger in the back seat, the exposure to contaminants from the front is minimised.

What is important to note is that although the comfort conditions from the simulations are not ideal, with such high temperatures, both systems, case 1 and 2 are not meant to function

or operate alone. They are to be used in tandem with other cooling inlets to assist in the stabilisation of temperatures. The exploration of the PV cooling method via this comparison is meant to understand how effective and what type of improvements may be expected when it is adopted.

5.6.3 Energy usage and savings

The energy required by the air conditioning to remove the heat and moisture from the airspace can be estimated with a series of equations.

Enthalpy of moist air:

$$h_i = h_a + xh_w \quad (5-1)$$

where

h_i	: Total enthalpy at inlet or outlet
h_a	: Enthalpy of dry air
h_w	: Enthalpy of water vapour
x	: Mass fraction of water vapour

Enthalpy of Dry air:

$$h_a = C_{pa}T \quad (5-2)$$

where

C_{pa}	: Specific heat capacity of air at constant pressure
T	: Temperature of air

Enthalpy of water vapour, latent heat:

$$h_w = C_{pw}T + h_{we} \quad (5-3)$$

where

C_{pw}	: Heat capacity of water
h_{we}	: Heat capacity of water at 0°C

The energy required at the condenser would be the difference in enthalpy at outlet and the enthalpy at inlet:

$$E = \Delta h = h_o - h_i \quad (5-4)$$

However the energy usage and savings cannot be calculated precisely with the 2 cases at the moment. This is due to the following reasons:

1. The cooling of the entire cabin has not been done for case 2 and case 3. Thus the temperatures obtained at the cabin are on the extremely high side.
2. Case 2 and case 3 both use the same amount of mass flow rate and inlet temperatures
3. Energy required to remove the latent heat in the space would not be accurate as the manikins are sweating quite drastically due to the extreme temperatures in the cabin.

In an actual properly designed cabin, little or no sweating should occur for the manikin.

Due to the above reasons, the values obtained for the energy usage would be of no significance. Using the current cases of case 1 and 2, it would be quite clear that the PV system would have the advantage over the conventional system. Having less moisture in the air, the latent heat would be lower and thus result in lower energy requirements. Moreover, the temperature of the inlet and the mass flow rates are the same for both cases, thus resulting in only a small different in enthalpy for those values.

However with that said, it is likely possible that the PV seat has a high potential for better energy savings. This is due to the fact that if the comfort level was kept at the same score, the PV system would most certainly require less energy (as a not so low inlet temperature or a lower inlet mass flow rate would be required) to cool the passenger in the cabin space.

Moreover, the PV system would most certainly have a reduced energy requirement when the vehicle is not a full capacity, restricting and cooling only the regions in the cabin with passengers present.

6 Conclusions

With the understanding of the heat load behaviour and expected heat sources that occurs in a vehicle cabin, it was found there are numerous methods to control the incoming heat. Changes in geometry of the car body, surface radiation properties of the car body materials, windows geometry and material properties and thermal capacity of the materials used in the car such as seat, dashboard and those used for the car body can help in the reduction of thermal heat load. The effective methods are usually those which are able to minimise the exposure of radiation coming into the cabin. As such focusing on the geometrical angles of the vehicle design and taking note of the materials used in the vehicle would assist in reducing the air conditioning energy requirements.

For every phase of the vehicle, from parking to cooling down to steady state cruising, the cooling down will require the most energy. Depending on the usage of the vehicle, if it is highly utilized for driving long distances frequently, steady state cooling would be a vital consideration for energy consumption. This air conditioning system should be designed to cope with such loads to give the optimal efficiency.

The next important energy saving source would be the design of the air distribution system in the cabin. The air distribution would need to be able to fulfil the specifications set out for the design of this system. Ultimately, the objectives of comfort and energy would have to be met. HVAC applications have been moving toward personal ventilation (PV) type of ventilation. As of now, such ventilation has been researched in buildings and aircraft cabin.

As electric vehicles become more widespread and with the urgent need to reduce the energy usage of air conditioning system, PV system have the potential to allow for energy savings as shown for buildings (Schiavon et al., 2010). With a good cabin distribution, an ideal compromise between energy savings and thermal comfort would be found. As PV would require more control in the methods of air delivery, secondary research can look into the development and setup of the air conditioning system and HVAC box to be able to supply the air flow rates and temperatures required.

With the existing conventional air conditioning and PV system compared, there are various advantages of a PV system, and various improvements and investigations which need to be done for this design to be realised.

Through the simulations, it was found that PV system resulting in better comfort, lower cross contamination risk of air breath out of different passengers, potential lower energy usage, and increased flexibility in cooling methods. By employing a top down cooling approach, it resulted in the following advantages:

1. The main reason for adopting a PV seat rather than having inlet situated directly at the top of the ceiling is that in a small vehicle like car, the heat loads from some surfaces like the dashboard is able to affect the airflow greatly. The warmer air from the dashboard would be able to push against the low velocity inlet air, thus changing its intended flow path. The use of the PV seat allows the inlet to be situated away from the path of the warmer air, reducing the disturbance of the warmer air rising in

affecting the cooler inlet air. This means that with the PV seat velocities as low as 0.3 m/s would not be affected by the warmer air rising to the ceiling of the vehicle.

2. Case 3 simulation has shown that the airflow is conformed to the path which is desired. The flow enters the cabin air space via the inlet at the cabin roof, and is directed to the top of the passenger. The air flow follows the contour of the person from the hand down to the core body regions of the chest and abdomen, followed by the thighs.

The PV seat results in a full body cooling of the entire passenger. However with the current design, some part of the body has little air blown towards it, such as the lower arms. This results in the arms feeling quite warm with a sensation value greater than 3 and a comfort value close to 4. The most of the vital regions of the body varied from -1 to 2 feeling cool to little warm. The discomfort of the arms is due to high solar radiation load comes through the windscreen and lack of direct cooling. Therefore this would be one point to be investigated.

Compared to the convective air conditioning with just mass flow rate for one person, the PV system has shown to be able to direct air to the core regions of the body. It is able to work with lower velocity air of only 0.3 m/s and yet have improved cooling effect having Zhang's comfort values of -1.34 meaning only slightly uncomfortable. High speed air might potentially result in a noisy cabin space and a more disruptive air flow. Using the Zhang's comfort index, the conventional air conditioning layout had a comfort value from -1.35 to -2 which is considered quite uncomfortable. The comfort value varies so much due to the sweat produced to stabilise the body temperature. This would mean the PV system had an improvement of about 0.7% to 50%.

3. Although it was not the main objective to study the effects of cross contamination of air from different passengers, it was possible to draw an understanding that the air flow would minimise cross contamination. One objective would be to ensure that particles from one person do not go into the breathing space of the other. The PV system does that by ensuring that the inlet is very close to the breathing zone of the person. With the use of this method, it ensures that recirculated air from the cabin has little or no chance of reaching the breathing zone of the person. The effectiveness of this method can be seen from the simulations results.

The N_2 mass fraction simulated the spread of contaminants in the cabin, which can serve as a gauge for contaminants flow. The PV system spread of N_2 is very much restricted to the front region of the vehicle. This can be seen also from the streamlines of the air particles route. This is better than the conventional air conditioning which resulted in the spread of the contaminants (N_2 as an example) to the back seat region of the vehicle from the front. In fact taking a measurement of the breathing zone at the back seat showed that the PV system had 32% less N_2 by mass compared to the conventional air conditioning system.

4. Potentially, it would be possible to make improvements to the PV design with respect to the whole cabin cooling to ensure that the comfort value for the PV and conventional system are equivalent with a lower mass flow rate. This would be due to lower “wastage” of air blown into region with no passengers. PV system will bring the cooler air directly to the passenger first before the air moves away from the passenger and cooling the surroundings.
5. The PV seat system would also allow for new types of cooling methods, such cooling via top down or the use of different inlet at different locations. The seat shape and form can be adjusted to allow for different air flow paths that might enhance cooling effectiveness or energy use. Such a PV seat design also allows for an integrated cooling solution which would not need too much adjustment to existing vehicles for its use.

Nevertheless as shown in the simulations, the PV seat will need to operate in tandem with other cooling strategies in the cabin. For the simulation case 2, with only the PV system operating, the surrounding surfaces were very warm due to the lack of cooling. This resulted in long wave thermal radiation which is trapped inside the vehicle to be re-radiated out towards other surfaces like the passenger.

Apart from the PV seat design, the simulation also introduced a few conclusions on the comfort models. In a vehicle under a high solar load that might occur in tropical countries near the equator, the PMV and PPD showed that for a conventional air conditioning system the discomfort scale reached nearly the max values. It was also highly unlikely that the inlet conditions would result in such high discomfort too. Nevertheless, the PMV and PPD would be unable to give a figure for comparison for the case 2 and case 3 which was run with a much lower mass flow rate.

Likewise the ISO 14505-2 had a similar limitation as the PMV and PPD, giving a maximum value for comfort for case 1. All these comfort indexes, were not meant for environments with such high temperature difference between body parts. This was the reason why Zhang’s comfort index was used for this analysis.

To conclude the simulation analysis, the co-simulation between the CFD solver and the thermal solver was effective but time consuming. StarCCM, the CFD solver took considerable time to solve the fluid flow and energy equations due to the large mesh size. The transfer of data from the CFD solver and the thermal solver was also time consuming. Time was required to map the data point values from the volume mesh to the surface mesh used by the thermal solver. However, the comprehensive thermal model present in the thermal solver was able to give detail information on the manikin heat and mass transfer along with the comfort indices, which was extremely useful for analysis.

7 Future Work

The PV seat has been shown to be a viable alternative to the conventional air conditioning system for a vehicle. However, more research is needed in improving the design of the seat and the integration of the seat into the cabin air space cooling system.

The design of the seat currently can be better improved with the exploration of different nozzle placements and temperatures. The current top down approach with the inlets at the top of the person allows the most of the body to be cooled and be comfortable. However as shown in the simulation, some body parts like the lower arms were left out as the inlet were not able to direct air towards that direction. Therefore more work needs to be researched into for the development of a more effective PV layout that would have improved comfort and cooling effect.

An experiment on the seat development need to be conducted to verify the results of the simulation and understand what physical limitations might be present in order to proceed in this development work. Solar loading would need to be simulated in order to provide realistic and yet controlled conditions for the testing of this PV solution.

The integration scenario of a PV system to the overall cabin cooling strategy is also an important aspect of development work. As shown in the current simulations, the PV seat cannot work alone without other cooling methods to cool the surrounding surface. If these surfaces are not cooled the thermal radiation from these areas will be radiated towards the passenger and thus affecting the comfort of the passenger. Therefore one potential approach is to introduce some cooling methods at those areas to allow for the regulation of the temperature in those areas even when no one is present. This can begin with a full simulation of the entire cooling strategy in the cabin with all the passengers involved to understand how each passenger would feel with respect to the solar radiation direction.

For the actual product to be realised, the ducting of the air from the air conditioning system to the seat and other inlet ports need to be designed. This is essential as the ducting should have as little flow resistance as possible and thus reducing pressure losses. Moreover, if the air flows through the ducting at high speeds, noise might be an issue.

A more comprehensive PV design might be considered which allows for the adjustment of individual temperatures and air velocity for each person. This would need integration with the air conditioning system and ducting with climate controls to achieve this. However, this work will still require the development of the improved PV seat.

More research can also be done on the comfort models. The Zhang's sensation and comfort model is getting widely used as it is effective for the measurement of comfort with larger temperature difference between body parts. The older comfort models like ISO 14505-2 and PMV are more suited for environments with a consistent temperature zone around the person. This can hold true from most air conditioned environments, such as in the office or home. However for a vehicle with high radiation effects and high loading, the comfort scenario will be different with the change in the heat loading exposure. Work can be done to verify if the Zhang's comfort models are accurate for all temperature ranges, or can it be better refined for some scenarios.

8 References

- Bolashikov, Z. D., & Melikov, A. (2009). Improved of Personalized Ventilation by Control of the Convection Flow around Occupant Body, *115*.
- Bolashikov, Z., Melikov, A., & Krenek, M. (2010). Control of the Free Convective Flow around the Human Body for Enhanced Inhaled Air Quality : Application to a Seat-Incorporated Personalized Ventilation Unit, *16*(March), 161–189.
- Chan, a. (2003). Indoor–outdoor air quality relationships in vehicle: effect of driving environment and ventilation modes. *Atmospheric Environment*, *37*(27), 3795–3808. doi:10.1016/S1352-2310(03)00466-7
- Chao, C. Y., & Wan, M. P. (2004). Experimental study of ventilation performance and contaminant distribution of underfloor ventilation systems vs . traditional ceiling-based ventilation system. *Indoor Air*, 306–316. doi:10.1111/j.1600-0668.2004.00248.x
- Dusan, F. (1998). *Dynamic Simulation of Human Heat Transfer and Thermal Comfort* (p. 237). Leicester and Stuttgart.
- Dusan, F., Lomas, K. J., & Stohrer, M. (1999). A Computer Model of Human Thermoregulation for a wide range of Environmental conditions: The Passive System. *Journal of Applied Physiology*, *87*, 1957–1972.
- Dusan, F., Lomas, K. J., & Stohrer, M. (2001). Computer prediction of human thermoregulatory and temperature responses to a wide range of environmental conditions. *International Journal of Biometeorology*, *45*(3), 143–159.
- Farrington, R., & Rugh, J. (2000). Impact of Vehicle Air- Conditioning on Fuel Economy , Tailpipe Emissions , and Electric Vehicle Range. In *Earth Technologies Forum* (p. 9).
- Fiala, D., Lomas, K. J., & Stohrer, M. (2001). Computer prediction of human thermoregulatory and temperature responses to a wide range of environmental conditions. *International Journal of Biometeorology*, *45*(3), 143–59. Retrieved from <http://www.ncbi.nlm.nih.gov/pubmed/11594634>
- Foda, E., Almesri, I., Awbi, H. B., & Sirén, K. (2011). Models of human thermoregulation and the prediction of local and overall thermal sensations. *Building and Environment*, *46*(10), 2023–2032. doi:10.1016/j.buildenv.2011.04.010
- Foda, E., & Sirén, K. (2011). A new approach using the Pierce two-node model for different body parts. *International Journal of Biometeorology*, *55*(4), 519–32. doi:10.1007/s00484-010-0375-4
- Geiss, O., Tirendi, S., Barrero-Moreno, J., & Kotzias, D. (2009). Investigation of volatile organic compounds and phthalates present in the cabin air of used private cars. *Environment International*, *35*(8), 1188–1195. doi:10.1016/j.envint.2009.07.016

- Ghosh, D., Wang, M., Wolfe, E., Chen, K., Kaushik, S., & Han, T. (2012). Energy Efficient HVAC System with Spot Cooling in an Automobile - Design and CFD Analysis. *SAE International*, 01(0641). doi:10.4271/2012-01-0641
- Grivel, F., & Candas, V. (1991). Ambient temperatures preferred by young European males and females at rest. *Ergonomics*, 34, 365–378.
- Hagstrom, K., Sandberg, E., H.Koskela, & Hautalampi, T. (2000). Classification for the room air conditioning strategies. *Building and Environment*, 35, 699–707.
- Halvoňová, B., & Melikov, A. K. (2010). Performance of “ductless” personalized ventilation in conjunction with displacement ventilation: Impact of intake height. *Building and Environment*, 45(4), 996–1005. doi:10.1016/j.buildenv.2009.10.007
- Ho, S. H., Rosario, L., & Rahman, M. M. (2011). Comparison of underfloor and overhead air distribution systems in an office environment.pdf. *Building and Environment*, 46, 1415–1427.
- Huang, K. D., Tzeng, S., Jeng, T., & Chiang, W. (2006). Air-conditioning system of an intelligent vehicle cabin. *Applied Energy*, 83, 545–557. doi:10.1016/j.apenergy.2005.05.006
- Huizenga, C., Hui, Z., & Arens, E. (2001). A model of human physiology and comfort for assessing complex thermal environments, 36, 691–699.
- Ingram, A. (2012). BMW's New Climate Control uses Thermal Management Tech. Retrieved June 06, 2012, from http://www.motorauthority.com/news/1073169_bmws-new-climate-control-uses-thermal-management-tech
- International Standard. (2005). ISO 7730 3rd Edition.
- Jonsson, J. (2007). *Including Solar Load in CFD Analysis of Temperature Distribution in a Car Passenger Compartment* (p. 138). Sweden.
- Kaczmarczyk, J. (2003). *Human response to personalised ventilation*. Technical University of Denmark, Lyngby, Demark.
- Kaczmarczyk, J., Melikov, a, & Fanger, P. O. (2004). Human response to personalized ventilation and mixing ventilation. *Indoor Air*, 14 Suppl 8(Suppl 8), 17–29. doi:10.1111/j.1600-0668.2004.00300.x
- Kaczmarczyk, J., Melikov, a., Bolashikov, Z., Nikolaev, L., & Fanger, P. O. (2006). Human Response to Five Designs of Personalized Ventilation. *HVAC&R Research*, 12(2), 367–384. doi:10.1080/10789669.2006.10391184
- Khalifa, H. E., Janos, M. I., & Dannenhoffer, J. F. (2009). Experimental investigation of reduced-mixing personal ventilation jets. *Building and Environment*, 44(8), 1551–1558. doi:10.1016/j.buildenv.2008.11.006
- Knibbs, L. D., Morawska, L., & Bell, S. C. (2012, March). The risk of airborne influenza transmission in passenger cars. *Epidemiology and Infection*. doi:10.1017/S0950268811000835

- Kwon, C., Lee, C. W., & Foster, L. (2012). Development of an Energy-Saving Occupied-Zone HVAC System (OZ HVAC). *SAE International*. doi:10.4271/2012-01-0320
- Loomans, M. G. L. C. (1998). *The Measurement and Simulation of Indoor Air Flow* (p. 228). Netherlands.
- Melikov, a., Ivanova, T., & Stefanova, G. (2012). Seat headrest-incorporated personalized ventilation: Thermal comfort and inhaled air quality. *Building and Environment*, 47, 100–108. doi:10.1016/j.buildenv.2011.07.013
- Melikov, A. K. (2004). Personalized ventilation. *Indoor Air*, 12(Suppl 7), 157–167.
- Melikov, A. K. (2010). Performance of Ductless Personalized Ventilation in Conjunction with Displacement Ventilation :, 16(January), 75–95.
- Melikov, A. K. (2011). Advanced Air Distribution. *ASHRAE Journal*, 73–77.
- Melikov, A. K., Cermak, R., & Majer, M. (2002). Personalized ventilation: evaluation of different air terminal devices. *Energy and Buildings*, 34(8), 829–836. doi:10.1016/S0378-7788(02)00102-0
- Melikov, A. K., & Knudsen, G. L. (2007). Human Response to an Individually Controlled Microenvironment, 13(4), 645–661.
- Naíping, G., & Jianlei, N. (2008). Improvement of Facial Humidity and Reduction of Pollutant Inhalation in Commercial Aircraft Cabins via Application of Personalized Ventilation. *ASHRAE Transactions*. doi:SL-08-004
- Nilsson, H. O. (2007). Thermal comfort evaluation with virtual manikin methods. *Building and Environment*, 42(12), 4000–4005. doi:10.1016/j.buildenv.2006.04.027
- P.Zitek, Vyhldal, T., Simeunovic, G., Novakova, L., & Cizek, J. (2010). Novel personalized and humidified air supply for airliner passengers. *Building and Environment*, 45, 2345–2353.
- Park, S., Hellwig, R. T., Grün, G., & Holm, A. (2011). Local and overall thermal comfort in an aircraft cabin and their interrelations. *Building and Environment*, 46(5), 1056–1064. doi:10.1016/j.buildenv.2010.11.003
- Penot, F., & Meyer, P. (2000). THERMAL AND DYNAMIC STUDY OF A NEW CONCEPT OF HVAC SYSTEM USING AN ANNULAR AIR JET ., 1, 657–662.
- Penot, F., & Pavlović, M. D. (2010). Experimental Study of Non-Isothermal Diverging Swirling and Non-Swirling Annular Jets with Central Aspiration, 8(4), 347–357.
- Romana, F., Palella, B. I., Riccio, G., Ponte, V., Melillo, D., & Salerno, F. (2008). THERMODE 193 : AN ENHANCED STOLWIJK THERMOREGULATION MODEL OF THE HUMAN BODY, (September).
- Schiavon, S., Melikov, A. K., & Sekhar, C. (2010). Energy analysis of the personalized ventilation system in hot and humid climates. *Energy and Buildings*, 42(5), 699–707. doi:10.1016/j.enbuild.2009.11.009

- Sekhar, S. C. (2007). A Review of Ventilation and Air-Conditioning Technologies for Energy-Efficient Healthy Buildings in the Tropics. *ASHRAE Transactions*. doi:DA-07-045
- Sekhar, S. C., Gong, N., Tham, K. W., Cheong, K. ., Melikov, A. ., Wyon, D. P., & Ganger, P. O. (2005). Findings of Personalized Ventilation Studies in a Hot and Humid Climate. *HVAC&R Research*, 11(4), 603–621.
- Tham, K. W., & Pantelic, J. (2010). Performance evaluation of the coupling of a desktop personalized ventilation air terminal device and desk mounted fans. *Building and Environment*, 45(9), 1941–1950. doi:10.1016/j.buildenv.2010.01.019
- Van Hoof, J. (2008). Forty years of Fanger’s model of thermal comfort: comfort for all? *Indoor Air*, 18(3), 182–201. doi:10.1111/j.1600-0668.2007.00516.x
- Wargocki, P., Wyon, D. P., Sundell, J., Clausen, G., & Fanger, P. O. (2000). The Effects of Outdoor Air Supply Rate in an Office on Perceived Air Quality , Sick Building Syndrome (SBS). *Indoor Air*, 10, 222–236.
- Wong, L. T., & Chow, W. K. (2001). Solar radiation model. *Applied Energy*, 69(3), 191–224. doi:10.1016/S0306-2619(01)00012-5
- Yang, B., & Sekhar, C. (2009). Performance Evaluation of Ceiling Mounted Personalized Ventilation System, 115.
- Yang, B., Sekhar, C., & Melikov, A. K. (2010). Ceiling mounted personalized ventilation system in hot and humid climate—An energy analysis. *Energy and Buildings*, 42(12), 2304–2308. doi:10.1016/j.enbuild.2010.07.022
- Yang, J., Kaczmarczyk, J., Melikov, A. K., & Fanger, P. O. (2003). The impact of a personalized ventilation system on indoor air quality at different levels of room air temperature. In *Proceedings of Healthy Buildings 2003* (pp. 345–350). Singapore.
- Zhai, Z. J., Zhang, Z., Zhang, W., & Chen, Q. Y. (2007). Evaluation of Various Turbulence Models in Predicting Airflow and Turbulence in Enclosed Environments by CFD: Part 1—Summary of Prevalent Turbulence Models. *HVAC&R Research*, 13(6), 853–870. doi:10.1080/10789669.2007.10391459
- Zhang, H., Arens, E., Huizenga, C., & Han, T. (2010). Thermal sensation and comfort models for non-uniform and transient environments: Part I: Local sensation of individual body parts. *Building and Environment*, 45(2), 380–388. doi:10.1016/j.buildenv.2009.06.018
- Zhang, T. (Tim), Li, P., & Wang, S. (2012). A personal air distribution system with air terminals embedded in chair armrests on commercial airplanes. *Building and Environment*, 47, 89–99. doi:10.1016/j.buildenv.2011.04.035
- Zhang, T., & Chen, Q. (Yan). (2007). Novel air distribution systems for commercial aircraft cabins. *Building and Environment*, 42(4), 1675–1684. doi:10.1016/j.buildenv.2006.02.014

Appendix A Vehicle Components

Part	Material ID	Thickness (m)
rear_seat	65	0.04
	5	0.001
front_seat_left	65	0.04
	5	0.001
rear_seat_2	65	0.04
	5	0.001
front_right_seat	65	0.04
	5	0.001
Bonnet	1	0.0008
cabin_floor	8	0.006
	10	0.002
	67	0.02
	1	0.0008
	4	0.0006
dashboard_side	57	0.0008
	68	0.002
	22	0.005
	1	0.0008
engine_floor	1	0.0008
engine_sides	1	0.0008
bonnet_front	1	0.0008
rear_door	11	0.0008
	12	0.1
	13	0.004
	20	0.002
	1	0.0008
Roof	95	0.003
	94	0.001
	67	0.01
	92	0.04
	67	0.008
	1	0.0008
side_doors	11	0.0043
	12	0.1
	13	0.004
	20	0.002
	1	0.0008
sidea_and_rear_windows	24	0.004
side_walls	57	0.0008
	68	0.002
	22	0.005
	1	0.0008
windscreen	3	0.00217

	2	0.00076
	3	0.00217
windscreen_wall	57	0.0008
	68	0.002
dashboard_bottom	22	0.005
	1	0.0008
	6	0.008
	9	0.002
	1	0.0008
	6	0.006
dashboard_front	10	0.002
	64	0.0025
	65	0.008
	4	0.0012
dashboard_top	64	0.0025
	65	0.008
	4	0.0012
dasboard_front_2	64	0.0025
	65	0.008
	4	0.0012
dashboard_engine_side	10	0.002
	6	0.006
	1	0.0008
	9	0.002
	6	0.008
dashboard_topside	6	0.008
	9	0.002
	1	0.0008
	64	0.0025
	65	0.008
	4	0.0012

Appendix B Vehicle materials

Material	ID	Conductivity (W/mK)	Specific heat (J/kgK)	Density (kg/m3)
Metal sheet	1	46	460	7860
PVB (polyvinyl Butyral)	2	0.2	1200	1100
Greenglass-G90	3	0.9	800	2563
PVC (low density)	4	0.15	960	1100
Fabric	5	0.07	1300	430
PU-foam (low density)	6	0.033	1100	70
Carpet	8	0.07	1300	167
Fire Protection	9	0.16	1420	1000
PU+EPDM	10	0.16	1420	1000
Wood wool	11	0.15	1300	600
Air	12	0.027	1000	1.3
PVC	13	0.033	1100	140
Bitum	20	0.16	1420	1500
Fleece+board	22	0.04	1200	60
ESG-greenglass	24	0.9	800	2535
PVC (polyvinyl chloride)	57	0.15	960	1380
PU-foam (high density)	64	0.033	1100	700
PU-foam (medium density)	65	0.033	1100	175
PU-foam (low density II)	67	0.033	1100	30
PU-foam (high density II)	68	0.033	1100	1100
Airlayer	92	0.8	0.8	3.5
Fleece+CFK	94	0.04	1200	500
Décor (low density)	95	0.04	960	80

Appendix C Calibration case with Experimental VS Simulation Plot

Appendix D Plane slice for results plot for CFD modelling