# ANALYSIS OF HOT AIR HEAT EXTRACTOR FOR VEHICLE CABIN

SASHEENDRAN A/L KUPPUSAMY

A report submitted in partial fulfilment of the requirements
for the award of the degree of
Bachelor of Mechanical Engineering with Automotive

Faculty of Mechanical Engineering
UNIVERSITI MALAYSIA PAHANG

JUNE 2008

# CHAPTER 1

# INTRODUCTION

## 1.1 Background

The external powered car heat extractor is the ventilator that silently and efficiently rids heat out of vehicle cabin. While the windows remain securely closed, the solar powered cabin heat extractor eliminates hot air and odors from vehicle and replaces it with fresh air from the outside, ultimately reducing the interior temperature of the vehicle.

The heat extractor helps preserve and protect vehicle's interior, expensive stereo and electronic equipment from the damaging effects of extreme heat build-up. This heat extractor also assists in reducing the strain on air conditioning system during start-up. The system will be best installed to cars whose windows are tinted.

## 1.2 Problem Statement

Currently, there is no fixed type heat extractors specially designed for cars in the market. The available heat extractors in market are installed to the side window of car. All heat extractors that are available are non-fixed and it has to be removed whenever not in use. That is actually not convenient for drivers especially for people who do not park for long hours. By means, fixing and removing these portable type heat extractors are time consuming and does not guarantee security for cars. Therefore, a fixed type heat extractor is needed for cars.

A Proton Iswara Aeroback, has been chosen to carry out the project. The design of the heat extractor has to consider the whole range of heat flow in a closed boundary in order to provide a complete view of its performance.

The existing heat extractors are usually not properly refined and optimized to enable high flow rate of heat from interior of the car. In addition, the efficiency of the current heat extractor design is also an issue as it is designed for simplicity which only offers time consuming and lack in efficient heat flow performance.

## 1.3 Objectives

The objectives of the project are as follows:

1) To simulate the heat transfer between vehicle cabin and environment under direct sunlight.
2) To analyze a prototype of fixed car cabin heat extractor which uses external power source.

## 1.4 Scope

The scopes of the project are as follows:

1) Analysis on car cabin heat transfer.
2) Modeling Proton Iswara Aeroback's cabin using 3D modeling software.
3) Analysis on temperature distribution in the cabin.
4) Simulation prediction of temperature distribution in cabin under direct sunlight.

### 1.5 Flow Chart

**Design and develop a hot air extractor for vehicle cabin**

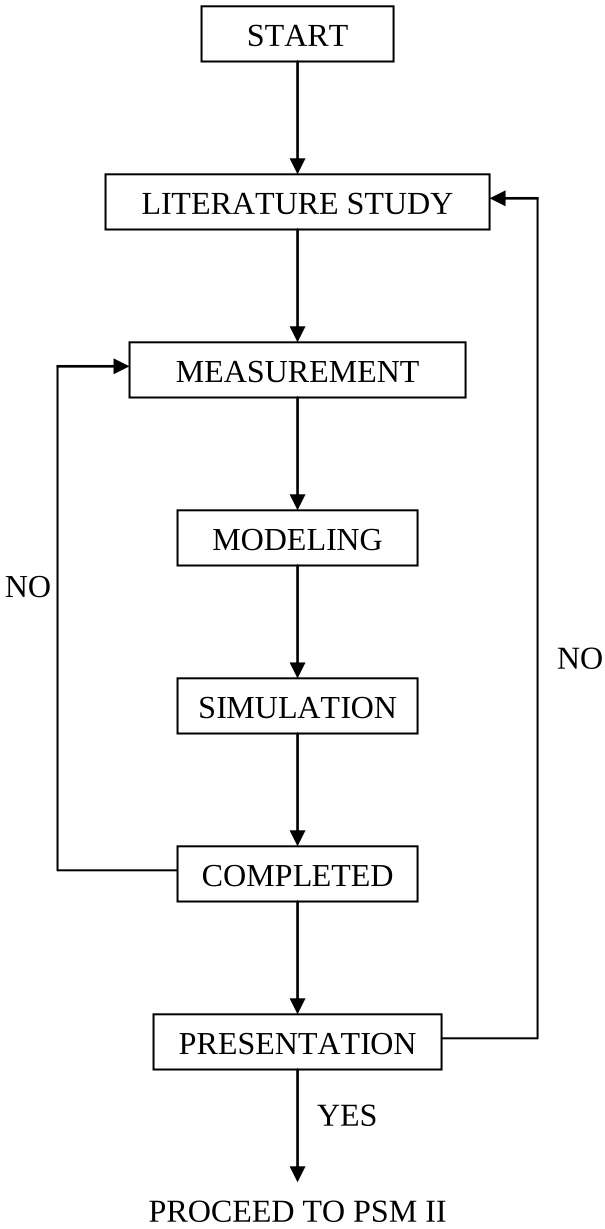


**Figure 1.1** PSM 1 Flow Chart

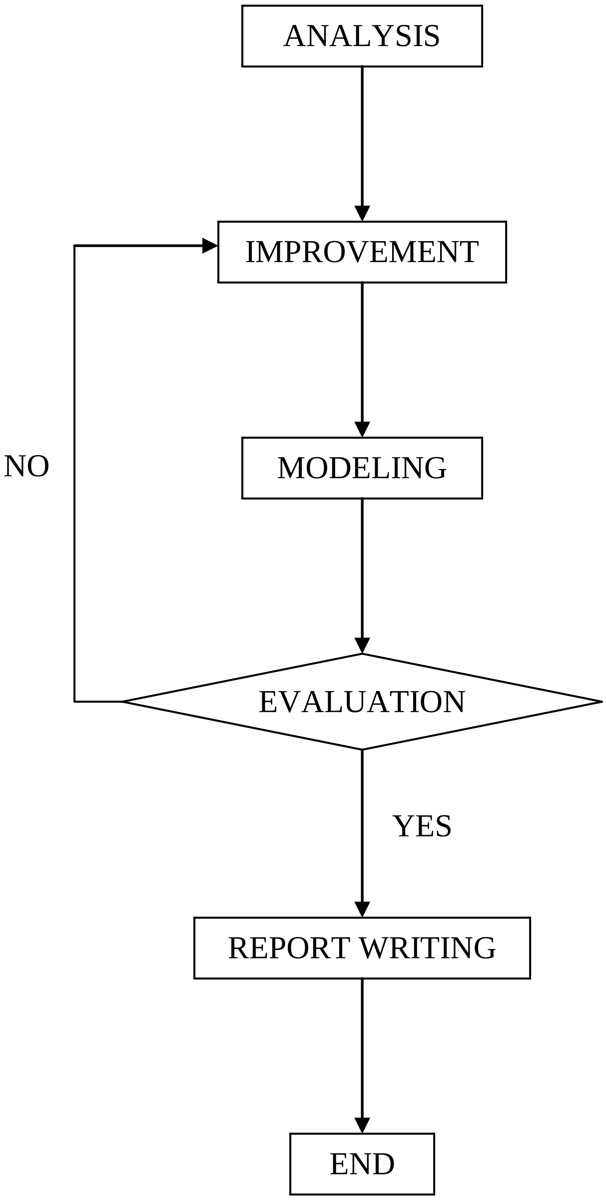


**Figure 1.2** PSM 2 Flow Chart

# CHAPTER 2

# LITERATURE REVIEW

## 2.1 Introduction to heat extractor

Vehicle cabin heat extractor is a device that uses fans to suck out air from inside the cabin to the environment. By doing this, it replaces the hot air with cooler air while odors brought out of the cabin at the same time. By meaning, the heat extractor cools and maintains the cabin temperature at optimum level while parked to ensure comfortness to the driver and passengers when they enter the car. Until today, only portable type of vehicle cabin heat extractors available in the market (Figure 2.1). The fixing and removing process of these heat extractors consumes time and not convenient for drivers who do not park car under direct sun light for hours. A fixed type heat extractor will be more convenient and efficient in removing hot air from the cabin. The available types in market are portable types and not within the reach for many people. Till now, only a few units are found to be sold through the internet. This is a small amount when compared to the number of cars in Malaysia. By means, the item is lacking in market.


**Figure 2.1** Available heat extractor in market(Moresales.com.my)

## 2.2 Heat transfer

Heat in a cabin develops 15% from road, 20% from engine and catalytic converter and 65% from sunlight (Tim Gilles, 2004). Heat loss or gain mainly occurs on a car cabin by three primary mechanisms:

a. Conduction
b. Convection
c. Radiation (Taniguchi, Yousuke, 2006)

### 2.2.1 Conduction

Conduction is the transfer of heat within an object or between two objects in contact which means it happens from warmer object to cooler object from molecule to molecule. The more dense air forces the lighter air to rise. For heat to conduct from one object to another, they must be in contact. (Tim Gilles, 2004)

### 2.2.2 Convection

Convective heat transfer occurs when a liquid or gas comes in contact with a material of a different temperature. Air in contact with the interior wall will warm, becoming less dense, and rise which means when air becomes warmer, it moves upward. The result is increased heat loss or gain. By stopping the air movement, convective heat loss will cease. (Tim Gilles, 2004)

### 2.2.3 Radiation

Radiation is the transfer of heat from one object to another by means of infra-red waves. Radiative heat transfer does not require that objects be in contact or that a fluid flow between those objects. Radiative heat transfer occurs in the void of space.

## 2.3 Fan as heat flow device

A Fan is a type of flow boundary condition. It is considered as an ideal device creating a volume or mass flow rate depending on the difference between the inlet and outlet static pressures averaged over the selected face (Waldron, Kenneth and Kinzel, Gory, 1999).

An Inlet Fan has a flow direction from fan to fluid. An Outlet Fan has a flow direction from fluid to fan.

Internal Fans have outlet (from fluid to fan) and inlet (from fan to fluid) faces. The static pressures needed at the faces for determining the fan flow rate are obtained during the flow calculation as values averaged over these faces (Peyret, R. and T.D. Taylor. 1990).


**Figure 2.2** Fan

**Table 2.1** Comparison between Various Fan Types (Eck Bruno, 1973)

| Fan types / Values | Backward, airfoil centrifugal | Forward curved centrifugal | Vane - axial | Propeller fan |
|---|---|---|---|---|
| **Fan total pressure Δp** | Higher Δp | Comparatively lower Δp | Higher Δp | Low Δp |
| **Flow rate** | All flow rates | Larger flow rate | All flow rates | Larger flow rate |
| **Fan power input** | Non-overloading | overloading | Non-overloading | Non-overloading |
| **Fan modulation** | Inlet vanes | Inlet vanes | Controllable pitch | |
| **Fan total efficiency** | 0.7 to 0.86 | 0.6 to 0.75 | 0.7 to 0.88 | 0.45 to 0.6 |
| **Sound power level** | Lower, higher dB at low frequencies | Medium, higher dB at low frequencies | Medium, difference of dB values small at various frequencies | Higher, Higher dB at high frequencies |
| **Airflow direction** | 90˚ turn | 90˚ turn | Parallel to axle greater | Parallel to axle |
| **Volume and weight** | Greater | Less | Greater | medium volume, low weight |
| **Initial cost** | Higher | Medium | Higher | Low |
| **Application** | Large HVAC&R system | small HVAC&R system | Large HVAC&R system | high-volume flow exhaust system |

Table 2.1 shows that axial fan has the properties that required to be used as heat extractor for a vehicle cabin where it indirectly recommends the axial fan usage for heat extraction. This is because axial fan has higher total pressure development

and performs well at large and small flow rates. Besides that, axial fan has very small sound power level which means it is less noisy or noiseless than other fan types. Axial fan also supports large HVAC system cooling (Komoriya T, 1989).

### 2.3.1 Axial Fan

Nowadays, axial fans are widely used in cooling systems. This is because an axial fan does not have positive or negative displacements. It cools systems continuously and the volume flow rate varies according to the system condition. The axial fan would be the best option for cooling as it an independent construction with minimum losses (Eggleston, D.M. and F.S. Stoddard. 1987).

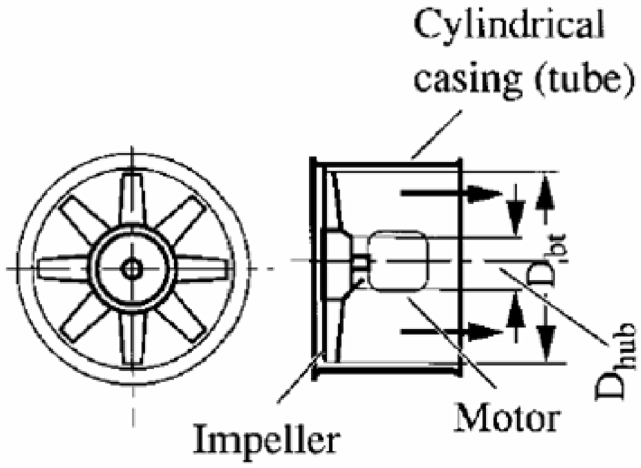


**Figure 2.3** Typical axial fan components (*Wang, S. K., 2001*)

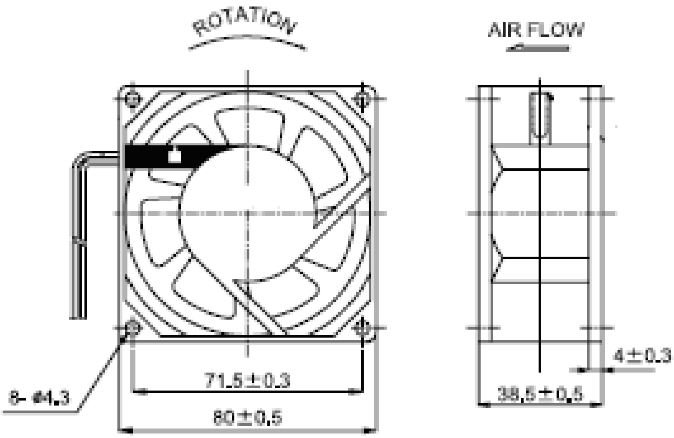


**Figure 2.4** Sunnon 80 x 80 x 38mm internal axial fan construction

## 2.4 Fan Related Calculations

Fans of the same basic design operate theoretically in accordance with certain fan laws. In practise, these laws do not apply exactly because of design considerations and manufacturing tolerances, but they are useful in estimating approximate outputs of similar fans of different diameters and speeds as applied to normal ventilation work, and can be summarised as follows (John P. Rugh, 2006):

a. Volume of air flow varies as fan diameter and as rpm.
b. Pressure developed varies as fan diameter and as rpm.
c. Power absorbed by the fan varies as fan diameter and as rpm.

These laws are most often used to calculate change in flow rate, pressure, and power of a fan when the size, rotational speed or gas density is changed (Barton, Lyndon, 1993).

### 2.4.1 Fan Velocity

Fan Velocity (m/s) = Radius of Fan (m) x Fan RPM

$$V = r\,\omega$$ (Myszka, David H. 2005)

### 2.4.2 Fan Mass Flow Rate

Pressure(kPa) x Velocity(m/s) = Mass(kg) xGas constant(kJ/kg.K) x Temperature(K)

$$PV = mRT$$ (Finnemore and B.Franzini. 2002)

$$P = mRT / V$$

$$P / RT = m / V = \rho$$ (air density, kg/m³)

Fan mass flow rate (kg/s) = air density (kg/ m³) x Fan area (m²) x Fan velocity (m/s)

$$M = \rho\, A\, V$$ (A. Çengel, Yunus and A. Boles, Michael. 2007)

### 2.4.3 System Resistance

The loss of pressure due to all of these sources, known as the system resistance, is for practical purposes proportional to the square of the velocity at the point of loss. Therefore, for a fixed system, it may be said that the pressure required to pass a given volume of air through the system will vary (Krieger, Malabar, FL. 1993).

Therefore, if it is required to double the air flow through a system, the fan must be capable of providing twice the volume flow rate at four times the original pressure and times the fan motor power (Ackers, P., W.R.White, A.J.Harrison and J.A.Perkins. 1978).

## 2.5 Role of Temperature, Time and Extractor Location in Cabin Heat Transfer

### 2.5.1 Temperature variations with time at different locations (without cooling)

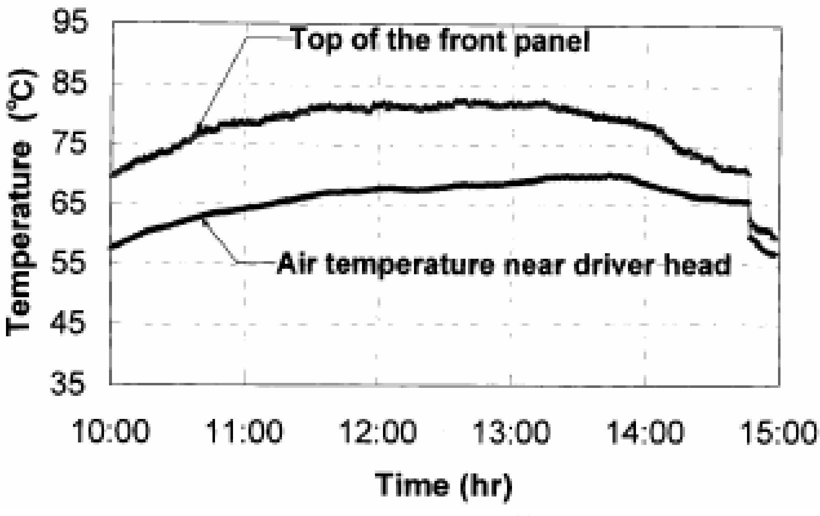


**Figure 2.5** Temperature variations with time at different locations (without cooling)

Figure 2.5 shows temperature variation of the front panel surface and air space temperature near drivers head without ventilation. While this experiment was carried out, the atmospheric temperature was 32˚C during 10:00am to 3:00pm. Front panel surface had the maximum temperature of car cabin which is 83˚C while the maximum air space temperature near driver head was found 67˚C (Khan,M.U, 2002).

### 2.5.2 Temperature variations with time at different flow rates (m³/hr)

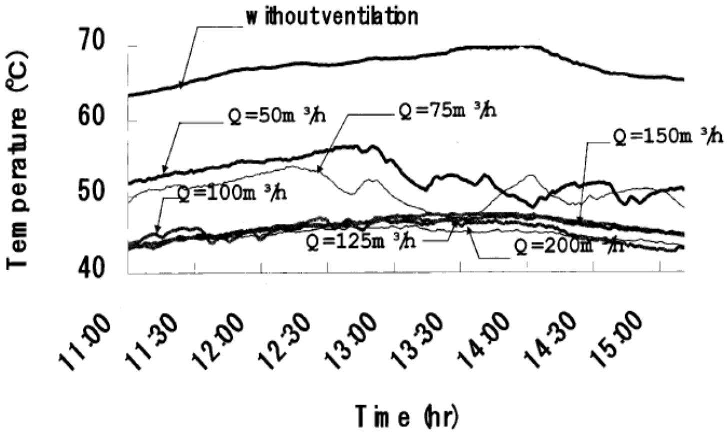


**Figure 2.6** Temperature variations with time at different flow rates (m³/hr)

Influence of air flow rate effects temperature variation near drivers head. This is shown in Figure 2.6. Temperature around 55˚C can be suppressed at air flow rate of 50 and 75m³/hr while airflow rates of 100, 125 and 200m³/hr can suppress temperature below 50˚C. This figure suggests air flow rates of 100m³/hr will be sufficient to mitigate temperature within comfortable range (Khan,M.U, 2002).

### 2.5.3 Effect of different location of air inlet and ventilation methods on ventilation

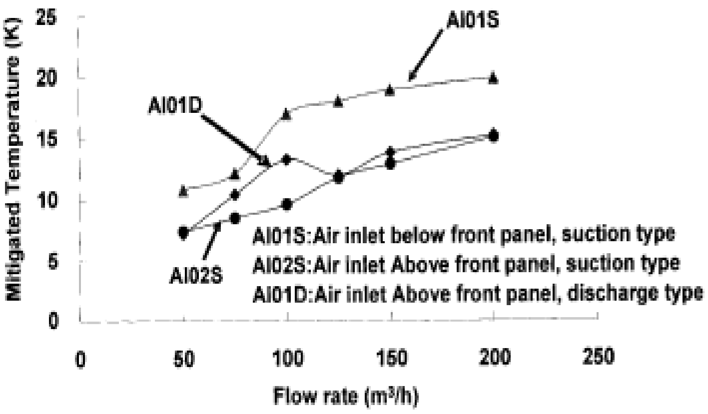


**Figure 2.7** Effect of different location of air inlet and ventilation methods on ventilation

Temperature mitigation at different air inlet locations and different ventilation methods of air space temperature near drivers head shown in figure 2.7. At air inlet below front panel and suction type ventilation, 12K of temperature mitigated for airflow rate 50 and 75$m^3$/hr. About 20K temperature mitigated at 100, 125, 150 and 200$m^3$/hr airflow rate. Air inlet above front panel and suction type ventilation, less than 10K temperature mitigated for airflow rate 50 and 75$m^3$/hr. Above 10K temperature mitigated for airflow rate of 100, 125, 150 and 200$m^3$/hr. Air inlet above front panel and discharge type ventilation mitigated 7K and 10K temperature for airflow rate 50 and 75$m^3$/hr respectively. About 15K temperature mitigated for airflow rate of 100, 125, 150 and 200$m^3$/hr. In conclusion, this analysis shows that air inlet below front panel with suction type ventilation have better temperature mitigation (Khan,M.U, 2002).

# CHAPTER 3

# METHODOLOGY

## 3.1 Introduction

Methodology need to be set first to achieve the objectives of the project. This is also important to ensure that the project is completed within the given time. This chapter lists all the relevant methods that needed during the two semesters of this final year project.

Gantt chart is recommended to be created to determine all works with duration of time. This is to ensure all works were carried out and completed by the dateline.

## 3.2 Problem Solving

In this project, the main problem solving tool is the Computational Fluid Dynamics software (CFD), which is used to analyze and simulate the heat transfer and heat flow of the car cabin. Problem solving is done according to the methodology flow chart where the problem solving methods were well organized.

### 3.2.1 Literature Study

This project refers to the vehicle cabin of a passenger car. It is important to make a study on the basics of heat transfer and thermo fluid such as heat flow characteristics, heat extraction coefficient and other relevant requirements.

This part explains in detail about the design of the heat extractor and all necessities of heat extraction from the cabin.

The literature study is an on going process from the beginning to the end of the project. This will ensure latest information updated time to time.

### 3.2.2 Measurement

The car model is a Proton Iswara Aeroback which is converted to electric vehicle by substituting the internal combustion engine with an electric motor. Besides collecting the car’s overall dimension from its user manual, the cabin dimension was measured manually as it is not stated in the user manual. The manual measurement process was done using measuring tape, Vernier caliper, micrometer and some other measurement aiding tools. The measurements were double checked to avoid significant errors.

After a brief discussion with supervisor, the manually taken dimension is used in modeling the cabin using 3-D modeling software.

### 3.2.3 3-D Modeling Software

#### 3.2.3.1 3-D Cabin Modeling

By referring to the dimensions from user manual and manually taken measurements, it is transferred into a 3-D model using 3-D modeling software. The 3-D modeling software is also used to add fans to the firewall of the Iswara Aeroback in order to do CFD analysis and simulation.

Matching the dimensions of the Iswara Aeroback model with the actual perfectly is not possible as the thickness of the platform, roof and windows were assumed with the manual measurement.

This error in measurement is negligible as the project focuses on heat flow from inside the cabin through the fans, considering the cabin as a static boundary.

**3.2.3.2 Introduction**

3-D Modeling Software employs a parametric, feature-based approach to relating models and assemblies. Parameters refer to constraints or conditions whose values determine the size, shape, characteristics, and behavior of the model or assembly. Parameters can be either numeric, for example dimension values such as the diameter of a circle or the length of a line; or geometric, such as conditions like tangent, concentric, coincident, parallel, horizontal, and the like. Numeric parameters such as dimensions can easily be related to each other through equations to capture even the most complicated design intent.

**3.2.3.3 Features**

Features refer to the building blocks of the part. They are the shapes and operations that construct the part. Shape-based features would include slots, holes, bosses and the like that either add or remove material from the part. Shape-based features typically begin with either a 2D or 3D sketch. Operation-based features generally don’t have sketches. These types of features include operations like filleting, chamfering, shelling, or applying draft to a part.

**3.2.3.4 Sketch**

Building a model in 3-D Modeling Software usually starts with either a 2D or 3D sketch. The sketch consists of geometry such as lines, arcs, conics, and splines. Dimensions are added to the sketch to define the size and location of the geometry. Relations are used to define attributes such as tangency, parallelism, perpendicularity, concentricity, and such.

#### 3.2.3.5 Parametric Nature

The parametric nature means the dimensions and relations drive the geometry, not the other way around. The dimensions in the sketch can be controlled independently, or by relationships to other parameters outside the sketch. Another aspect of the feature based nature is it can roll back into the history of the part in order to make changes, add additional features, or change to sequence in which operations are performed.

#### 3.2.3.6 Parts

Parts are modeled following a feature-based approach. A sketch must be created first in order to define the primary geometry of the part.

#### 3.2.3.7 Drawing

Drawings can be created either from parts or assemblies. They are drawn automatically, just by clicking on the window that contains the part or assembly to draw. The drawing module includes most paper sizes and standards (ANSI, ISO, DIN, GOST, JIS, BSI and GB).

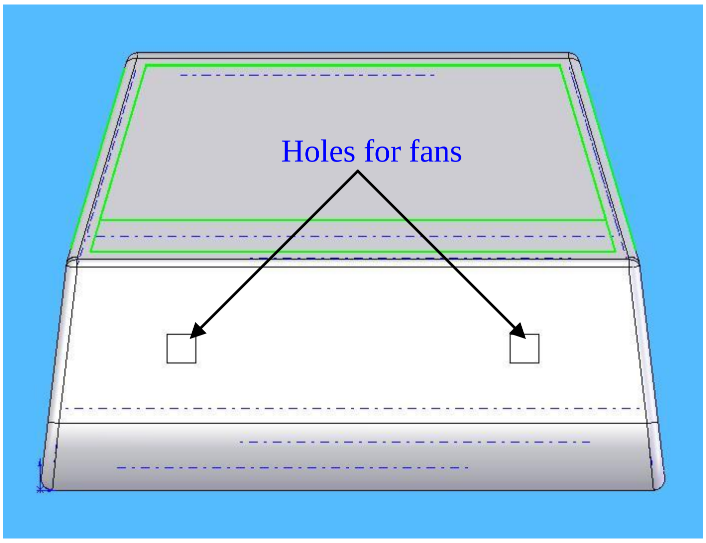


**Figure 3.1** 3D modeling Front view of HEV cabin

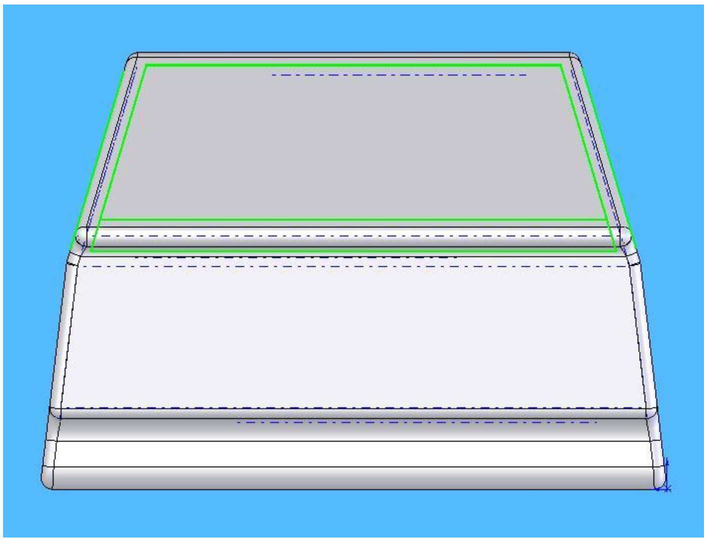

**Figure 3.2** 3D-modeling rear view of HEV cabin

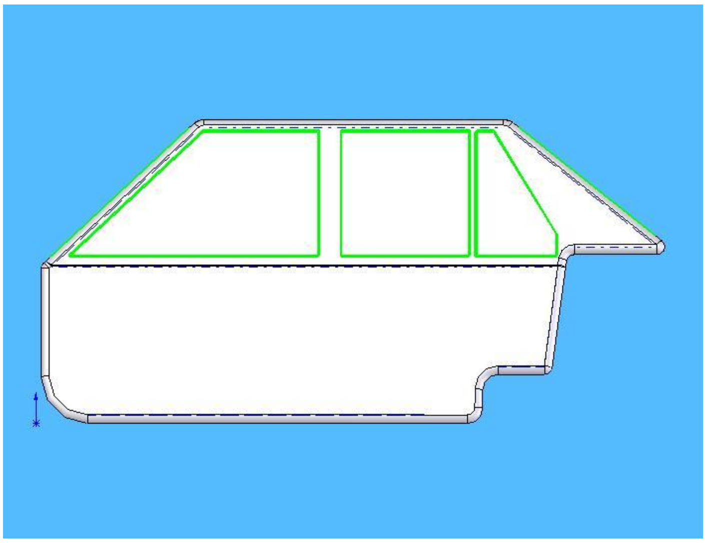

**Figure 3.3** 3D-modeling side view of HEV cabin

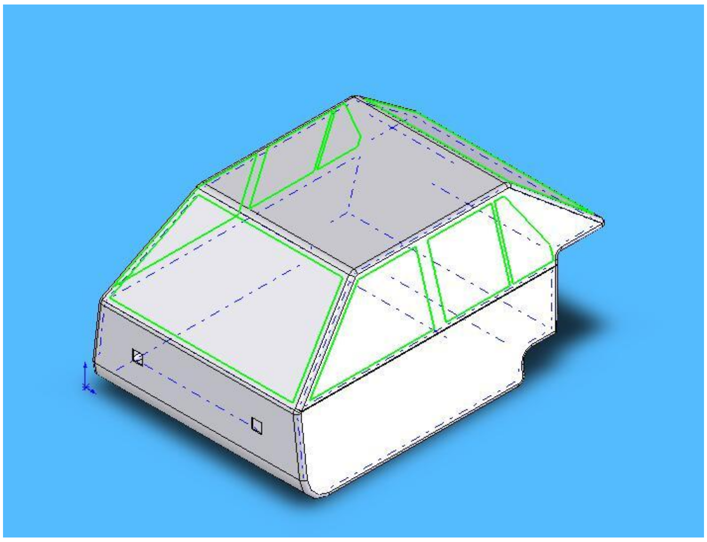

**Figure 3.4** 3D-modeling isometric view of HEV cabin

### 3.2.4 Computational Fluid Dynamics Simulation

Advanced computer simulation tools known as CFD (Computational Fluid Dynamics) have been employed in automotive development over the past few years. CFD simulations do not require real vehicles, CAD models are enough. These days extensive virtual simulations are performed on tens of computer processors even before the first prototypes are produced (Anderson, J.D. 1995).

CFD codes provide numerous possibilities of making the air-flow along and inside vehicles visible in any views and with any degree of detail. Nonetheless, expert users explore much more than just air-flow along the body where other parts of vehicle taken into consideration too. For example, the heat extraction inside the vehicle cabin to the environment. Many hours spent on guiding the air because the current vehicle cabins require efficient cooling and enough air to achieve maximum possible efficiency levels.

After the Iswara Aeroback cabin modeling completed, the next process is to transfer the model into CFD Simulation known as Computational Fluid Dynamics Simulation to make analysis on the heat flow rate and the temperature distribution in the cabin. By this, the heat flow rate to the environment from cabin can be set to achieve desired cabin temperature.

This CFD simulation is to determine the heat flow rate before and after the fans added to see the difference in cabin temperature. This CFD simulation does not need accurate dimensions as little change in measurement will not affect the result. By means, the cabin is considered as a closed boundary with two openings.

This simulation generally focuses on the temperature distribution in the cabin and the effect of heat flow rate to the cabin temperature after entering desired input. CFD simulation will show the most heated and less heated part of the cabin. Besides that, it also shows the heat flow inside the cabin.

During this simulation, all data were recorded and the analysis results were compared after all simulations done.

### 3.2.4.1 Meshing of the Cabin

Meshing done to ensure proper reading obtained from the simulation. Figure 3.5 and 3.6 shows the graphic of cabin meshing. This meshing contains 208757 fluid cells and 92527 partial cells.

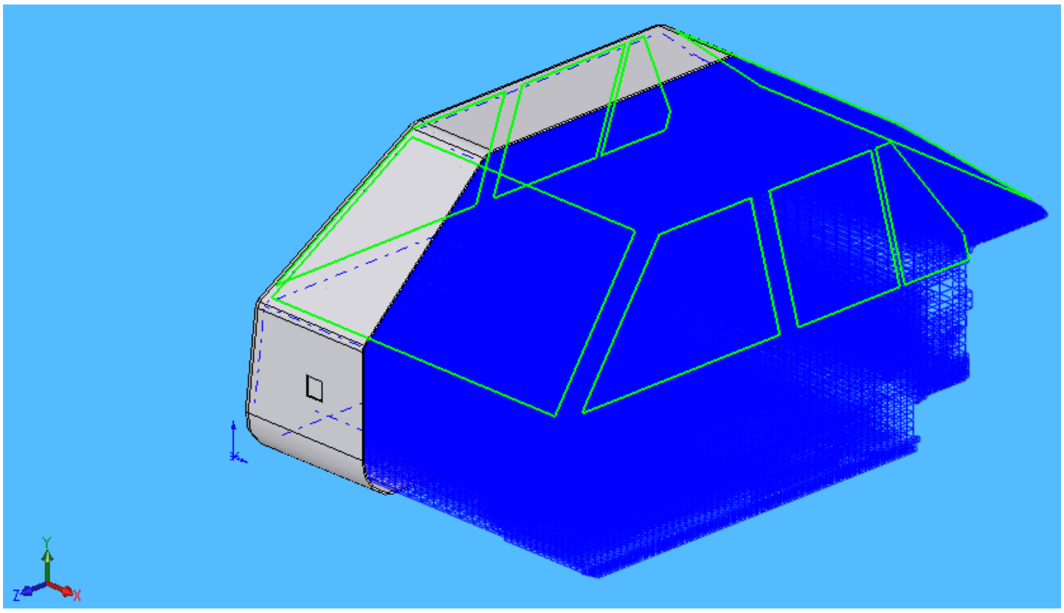

**Figure 3.5** 3D- mesh view

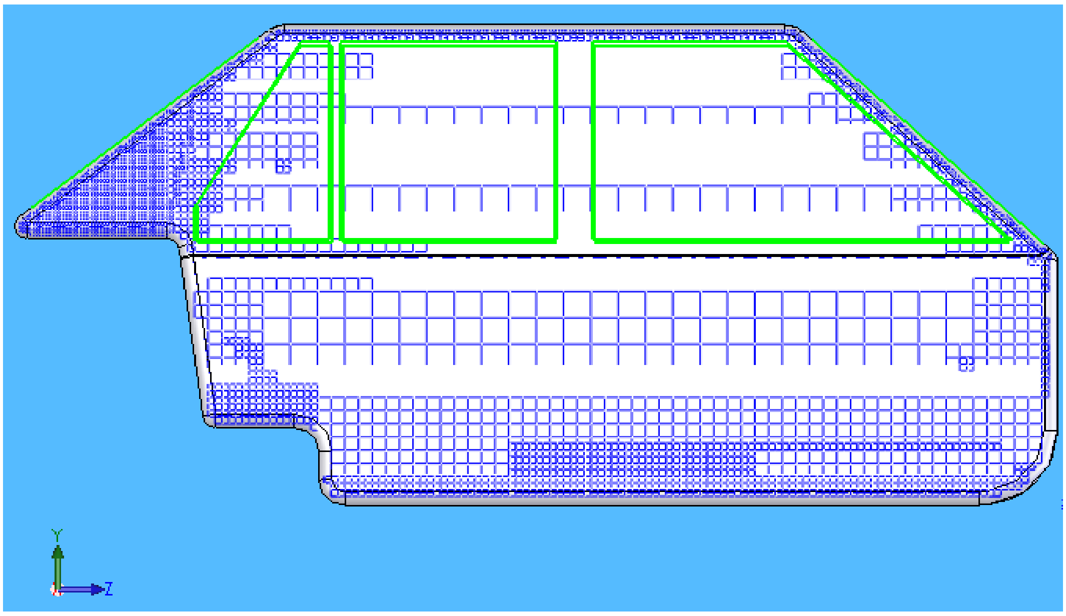


**Figure 3.6** Mesh side view

## 3.3 Temperature Reading

The temperature reading was taken from Proton Iswara Aeroback which has a closed cabin. The temperature for parked car under the sun for 30 minutes, 60 minutes and 5 hours measured using a room thermometer. The reading was taken at various places and the average temperature was set in the CFD software to obtain the temperature variation. Later, when the fans were added, the temperature mitigation was also obtained. In the figures below, the spots marked with red circles are the spots measured with room thermometer.

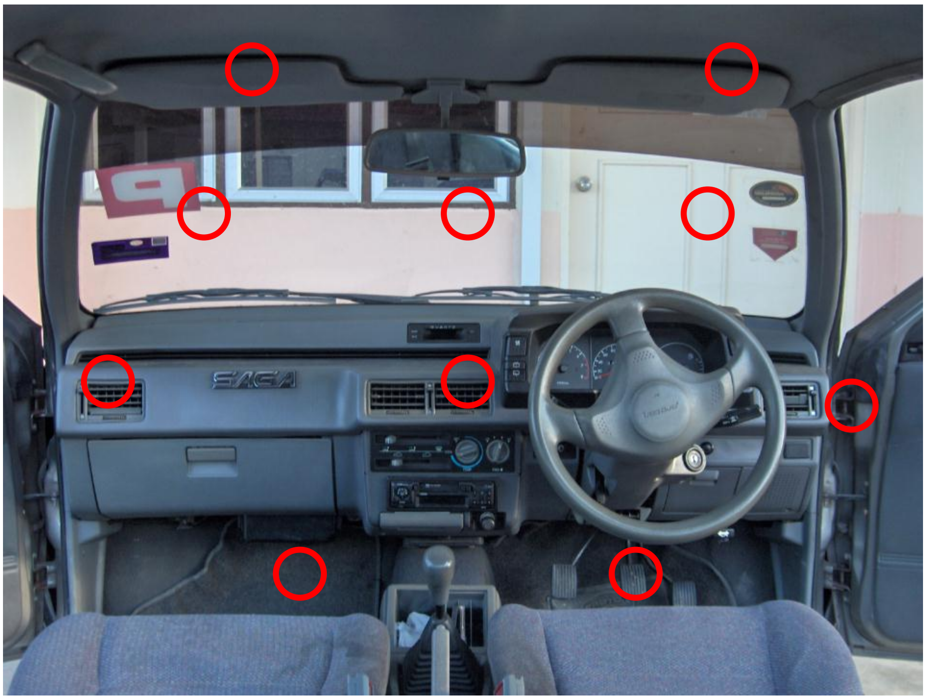

**Figure 3.7** Temperature measurement spots at the front panel

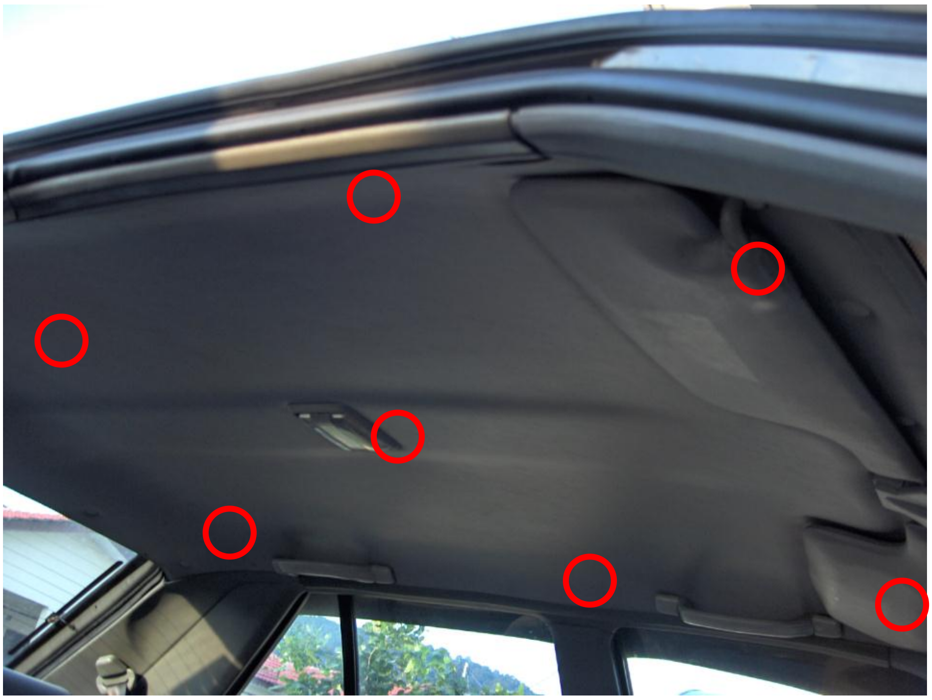

**Figure 3.8** Temperature measurement spots at the roof

**Figure 3.9** Temperature measurement spots at the front and rear side panels and windows

**Figure 3.10** Temperature measurement spots at the rear panel and window

**Figure 3.11** Temperature measurement spots at the front side panel and window

## 3.4 Fan RPM Reading

The fan RPM obtained easily by connecting the fan to the system fan socket at computer motherboard. The computer turned 'ON' after fixing the fan and the BIOS of the computer shows the RPM.

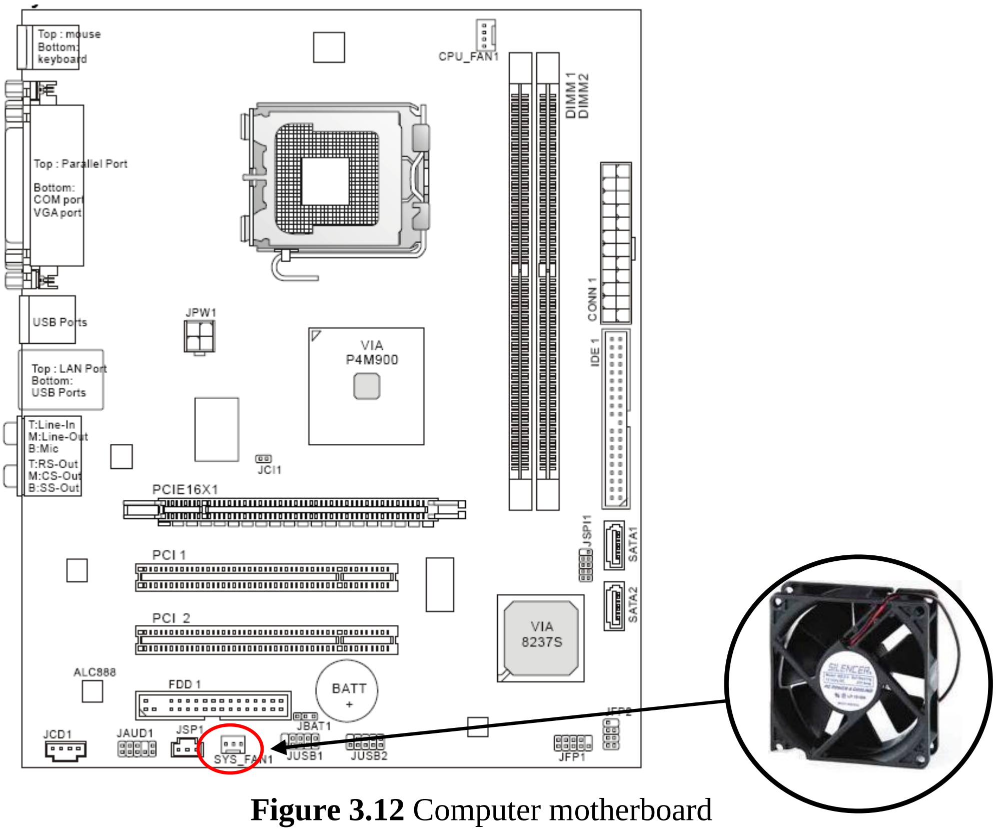


**Figure 3.12** Computer motherboard

## 3.5 Fan Mass Flow Rate

Fan mass flow rate is calculated in order to be used in CFD software as the software requires mass flow rate rather than velocity of the fan. The part that makes simulation making easy is that the software could calculate the air flow rate developed by the fan by just inserting the fan mass flow rate. A few calculations were done to obtain the fan mass flow rate at different air density as it varies by temperature. Below is the example calculation:

At 343K surrounding temperature and 1800 seconds time.

$$\omega(rad/sec) = \omega(rev/min) \times \pi /30$$

$$= 4300 \times \pi /30$$

$$= \mathbf{450.2949\ rad/sec}$$

$$V = r\,\omega$$

$= 0.03725 \times 450.2949$

$= \mathbf{16.773\ m/s}$

$$\rho = P / RT$$

$= 101.325 / 0.287 \times 343$

$= \mathbf{1.0293\ kg/m^3}$

$$M = \rho\, A\, V$$

$= 1.0293 \times \pi d^2 / 4 \times 16.773$

$= \mathbf{0.07526\ kg/s}$

# CHAPTER 4

# RESULT AND DISCUSSION

## 4.1 Introduction

The main objective of this project is to study the effect of heat flow rate and temperature distribution before and after the addition of the fans using the CFD simulation. Studies have been done on heat transfer which concluded that the heat transfer do play important part in cooling a vehicle cabin.

The main characteristic taken into consideration is the addition of fans to the vehicle firewall. By means, this is also one way to achieve ultimate cooling efficiency while maintaining the optimum temperature.

This simulation will determine whether there is any change or improvement before and after installing the fans to the vehicle firewall.

## 4.2 CFD Results

### 4.2.1 1800 seconds without heat extractor

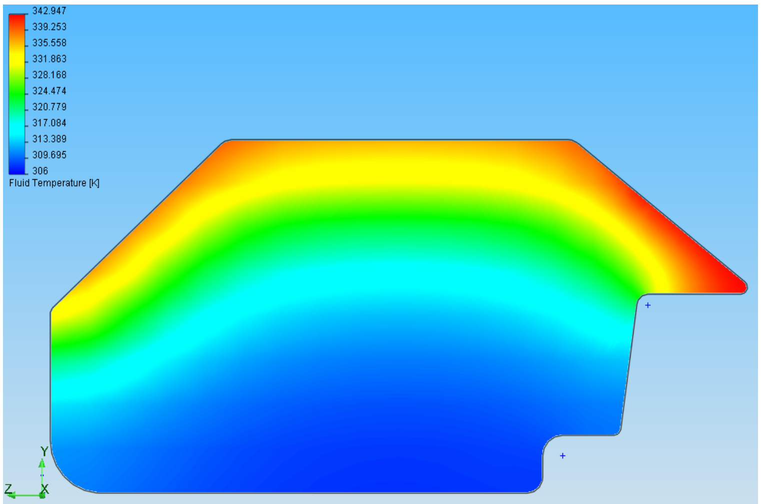


**Figure 4.1** Car cabin temperature distributions at 1800 seconds without heat extractor

Figure 4.1 shows the temperature distribution of car cabin parked under direct sunlight after 1800 seconds. Simulation shows minimum temperature of 310K at the floor and maximum temperature of 340K at the windscreen and rear window.

### 4.2.2 3600 Seconds Without Heat Extractor

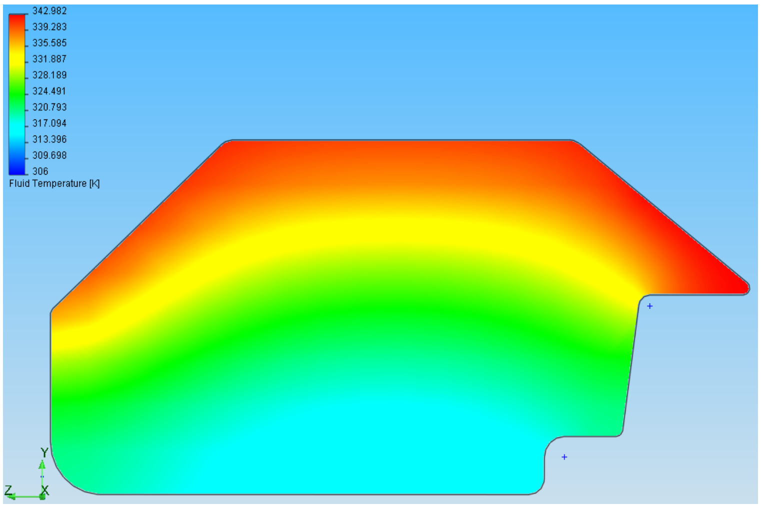


**Figure 4.2** Car cabin temperature distributions at 3600 seconds without heat extractor

Figure 4.2 shows the temperature distributions of car cabin parked under direct sunlight after 3600 seconds. Simulation shows minimum temperature of 315K at the floor and maximum temperature of 340K at the windscreen and rear window. The temperature distribution varies from top to bottom of the cabin.

### 4.2.3 18000 Seconds Without Heat Extractor

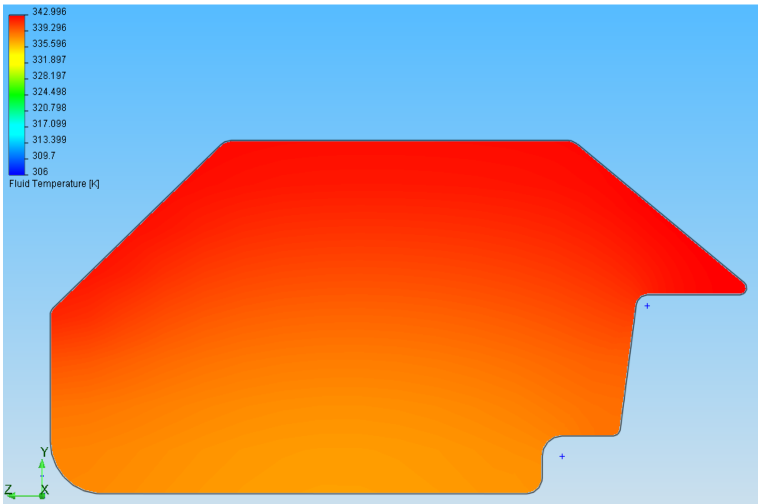


**Figure 4.3** Car cabin temperature distributions at 18000 seconds without heat extractor

Figure 4.3 shows the temperature distributions of car cabin parked under direct sunlight after 18000 seconds. Simulation shows minimum temperature of 337K at the floor and maximum temperature of 343K at the windscreen and rear window. The temperature distribution varies from top to bottom of the cabin.

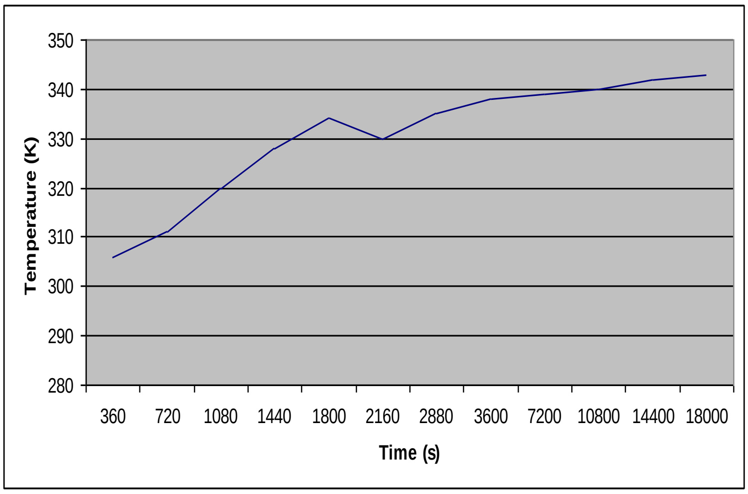


**Figure 4.4** Graph temperature versus time for without heat extractor

Graph shows that temperature rise is direct proportional to time. The temperature drops at 2160s to 330K because of pressure drop inside the cabin caused by pressure openings. Then, temperature starts to increase up to 343K.

### 4.2.4 1800 Seconds After Air Conditioning Without Heat Extractor

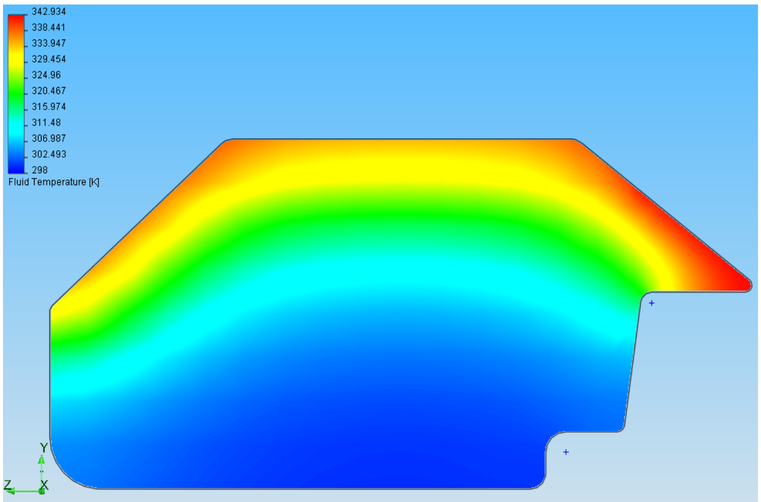


**Figure 4.5** Car cabin temperature distributions at 1800 seconds after air-conditioning without heat extractor

Figure 4.5 shows the temperature distributions of car cabin parked under direct sunlight after 1800 seconds. The simulation shows minimum temperature of 298K at the floor and maximum temperature of 333K at the windscreen and rear window. The temperature distribution varies from top to bottom of the cabin.

### 4.2.5 3600 Seconds After Air Conditioning Without Heat Extractor

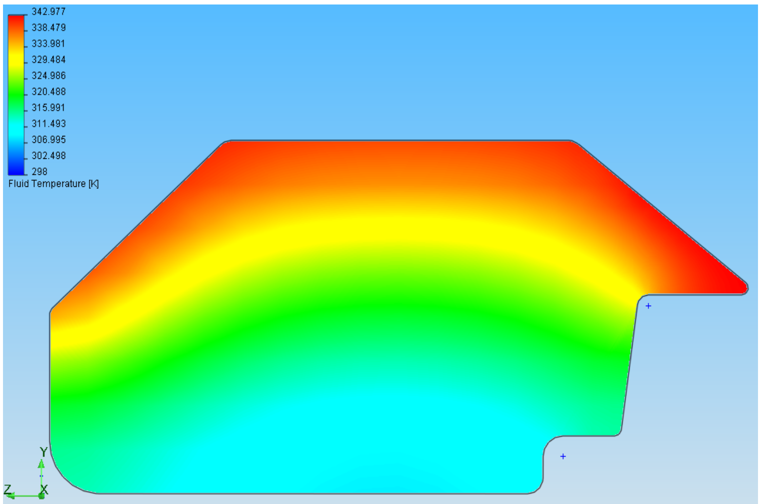


**Figure 4.6** Car cabin temperature distributions at 3600 seconds after air-conditioning without heat extractor

The figure 4.6 shows the temperature distributions of cabin parked under direct sunlight after 3600 seconds. The simulation shows minimum temperature of 310K at the floor and maximum temperature of 338K at the windscreen and rear window. The temperature distribution varies from top to bottom of the cabin.

### 4.2.6 18000 Seconds After Air Conditioning Without Heat Extractor

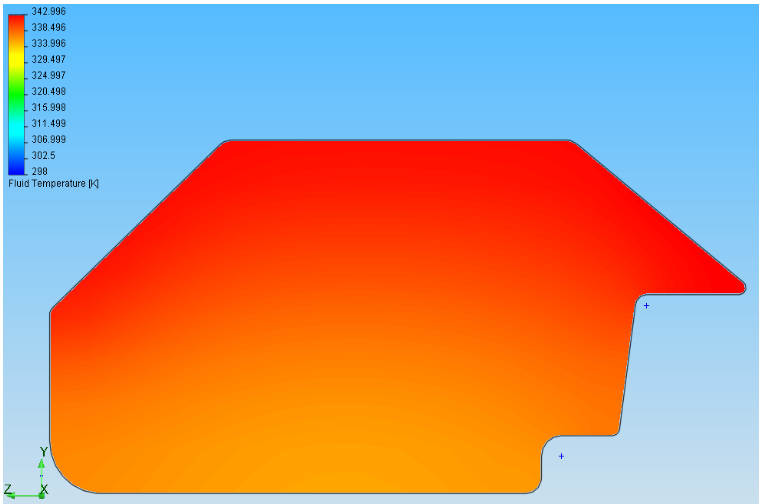


**Figure 4.7** Car cabin temperature distributions at 18000 seconds after air-conditioning without heat extractor

Figure 4.7 shows the temperature distributions of cabin parked under direct sunlight after 18000 seconds. The simulation shows minimum temperature of 335K at the floor and maximum temperature of 343K at the windscreen and rear window. The temperature distribution varies from top to bottom of the cabin.

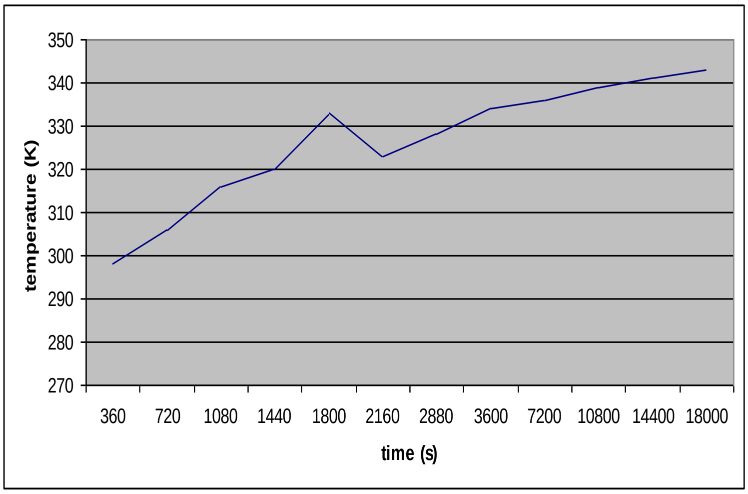


**Figure 4.8** Graph temperature versus time for after air-conditioning without heat extractor

Graph shows that temperature rise is direct proportional to time. The temperature drops at 2160s to 323K because of pressure drop inside the cabin caused by pressure openings. Then, temperature starts to increase up to 343K.

### 4.2.7 1800 Seconds After Air Conditioning and Tinted Without Heat Extractor

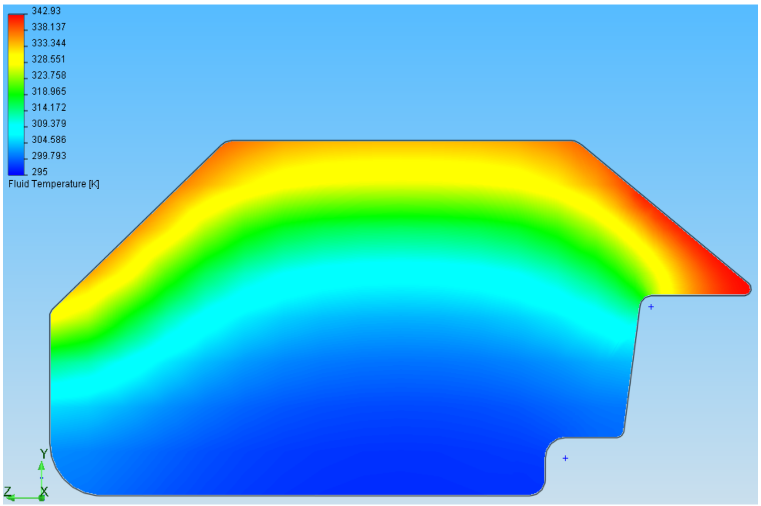


**Figure 4.9** Car cabin temperature distributions at 1800 seconds after air-conditioning and tinted without heat extractor

Figure 4.9 shows the temperature distributions of cabin parked under direct sunlight after 1800 seconds. The simulation shows minimum temperature of 300K at the floor and maximum temperature of 333K at the windscreen and rear window. The temperature distribution varies from top to bottom of the cabin.

### 4.2.8 3600 Seconds After Air Conditioning and Tinted Without Heat Extractor

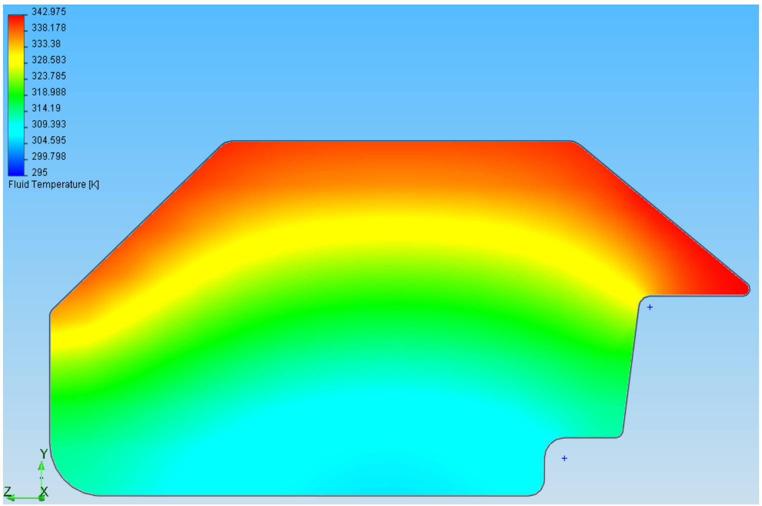


**Figure 4.10** Car cabin temperature distributions at 3600 seconds after air-conditioning and tinted without heat extractor

Figure 4.10 shows the temperature distributions of cabin parked under direct sunlight after 3600 seconds. The simulation shows minimum temperature of 310K at the floor and maximum temperature of 340K at the windscreen and rear window. The temperature distribution varies from top to bottom of the cabin.

### 4.2.9 18000 Seconds After Air Conditioning and Tinted Without Heat Extractor

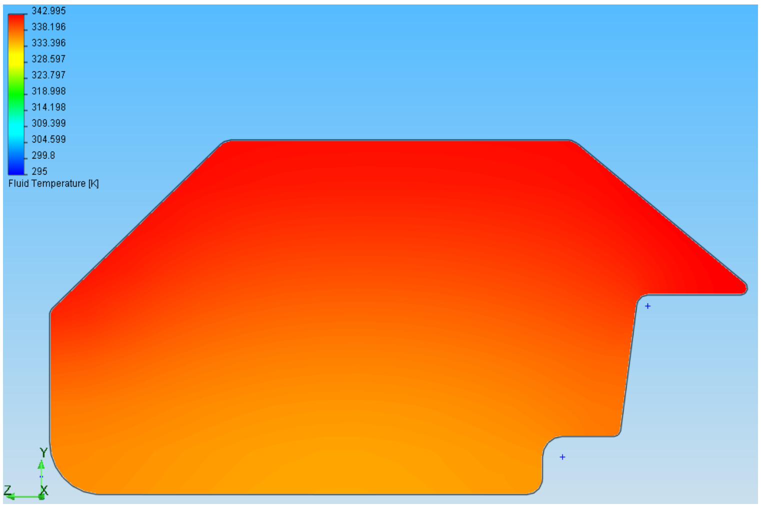


**Figure 4.11** Car cabin temperature distributions at 18000 seconds after air-conditioning and tinted without heat extractor

Figure 4.11 shows the temperature distributions of cabin parked under direct sunlight after 18000 seconds. The simulation shows minimum temperature of 335K at the floor and maximum temperature of 343K at the windscreen and rear window. The temperature distribution varies from top to bottom of the cabin.

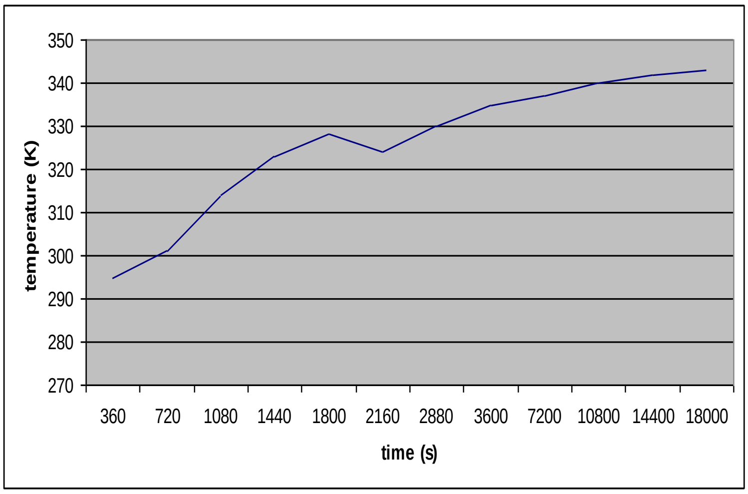


**Figure 4.12** Graph temperature versus time for after air-conditioning and tinted without heat extractor

Graph shows that temperature rise is direct proportional to time. The temperature drops at 2160s to 324K because of pressure drop inside the cabin caused by pressure openings. Then, temperature starts to increase up to 343K.

### 4.2.10 1800 Seconds Tinted Without Heat Extractor

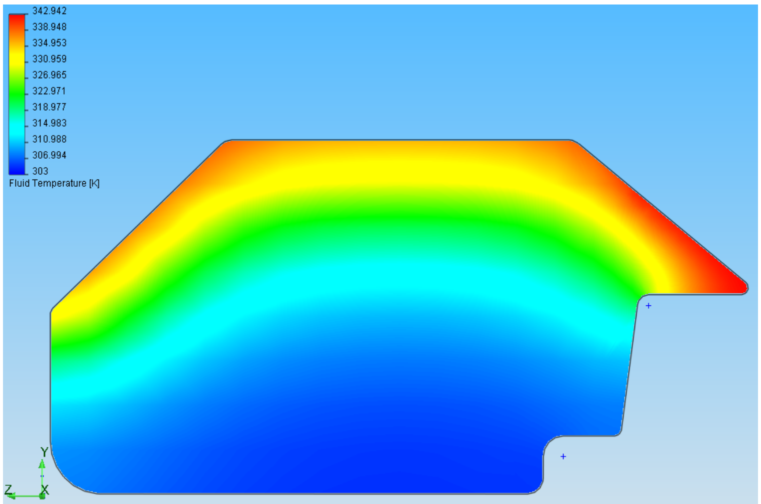


**Figure 4.13** Car cabin temperature distributions at 1800 seconds tinted without heat extractor

The figure 4.13 shows the temperature distributions of cabin parked under direct sunlight after 1800 seconds. The simulation shows minimum temperature of

303K at the floor and maximum temperature of 336K at the windscreen and rear window. The temperature distribution varies from top to bottom of the cabin.

### 4.2.11 3600 Seconds Tinted Without Heat Extractor

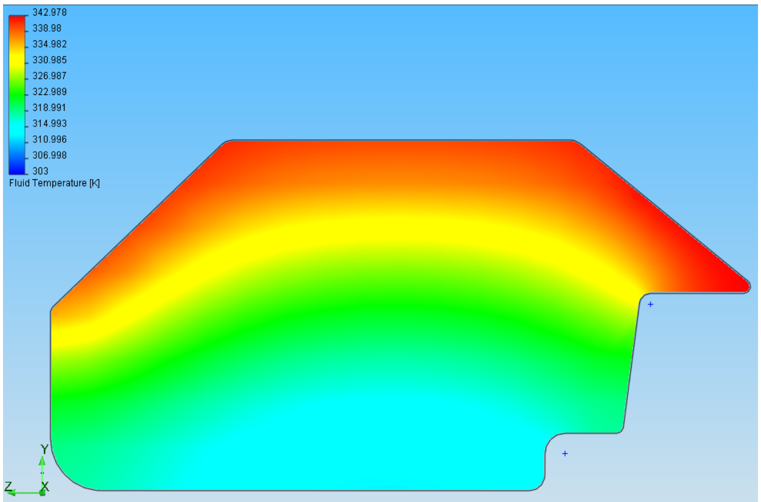


**Figure 4.14** Car cabin temperature distributions at 3600 seconds tinted without heat extractor

Figure 4.14 shows the temperature distributions of cabin parked under direct sunlight after 3600 seconds. The simulation shows minimum temperature of 315K at the floor and maximum temperature of 340K at the windscreen and rear window. The temperature distribution varies from top to bottom of the cabin.

### 4.2.12 18000 Seconds Tinted Without Heat Extractor

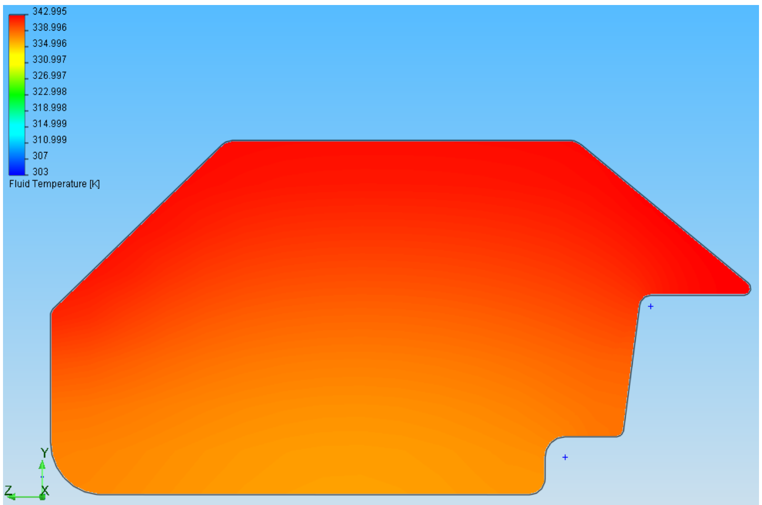

**Figure 4.15** Car cabin temperature distributions at 18000 seconds tinted without heat extractor

The figure 4.15 shows the temperature distributions of cabin parked under direct sunlight after 18000 seconds. The simulation shows minimum temperature of 336K at the floor and maximum temperature of 343K at the windscreen and rear window. The temperature distribution varies from top to bottom of the cabin.

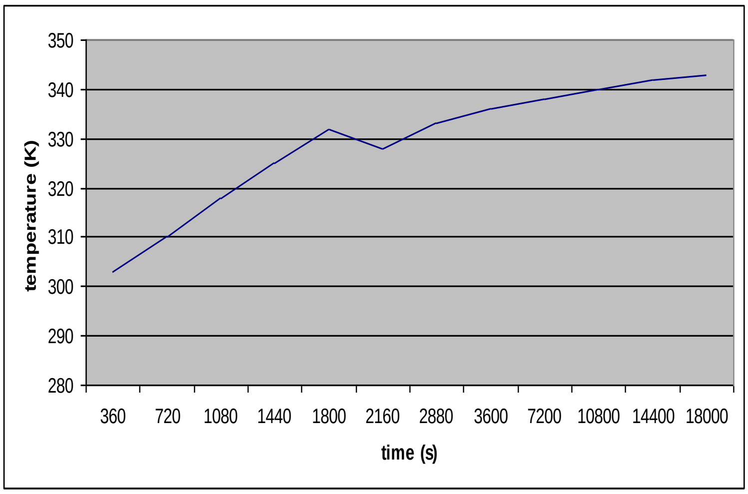


**Figure 4.16** Graph temperature versus time for tinted without heat extractor

Graph shows that temperature rise is direct proportional to time. The temperature drops at 2160s to 328K because of pressure drop inside the cabin caused by pressure openings. Then, temperature starts to increase up to 343K.

### 4.2.13 1800 Seconds With Heat Extractor

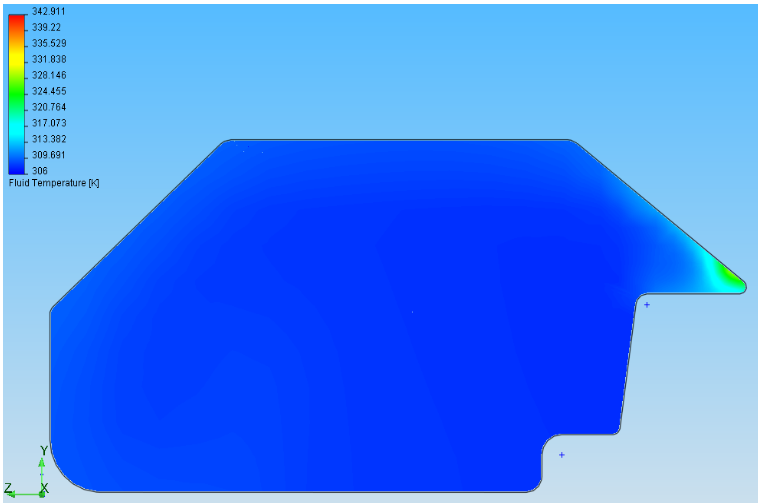


**Figure 4.17** Car cabin temperature distributions at 1800 seconds with heat extractor

Figure 4.17 shows the temperature distributions of cabin parked under direct sunlight after 1800 seconds with heat extractor. The simulation shows minimum temperature of 303K at most parts of the cabin and maximum temperature of 330K at the rear window. There is a small temperature difference at the rear window.

### 4.2.14 3600 Seconds With Heat Extractor

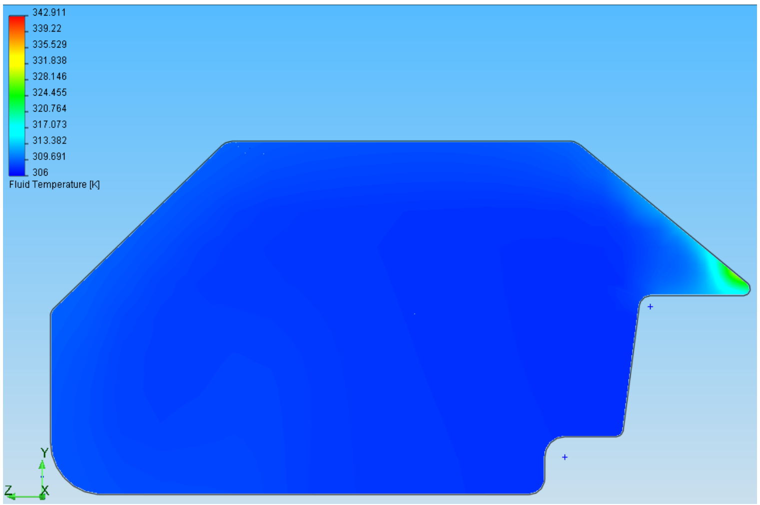


**Figure 4.18** Car cabin temperature distributions at 3600 seconds with heat extractor

Figure 4.18 shows the temperature distributions of cabin parked under direct sunlight after 3600 seconds with heat extractor. The simulation shows minimum temperature of 303K at most parts of the cabin and maximum temperature of 330K at the rear window. There is a small temperature difference at the rear window.

### 4.2.15 18000 Seconds With Heat Extractor

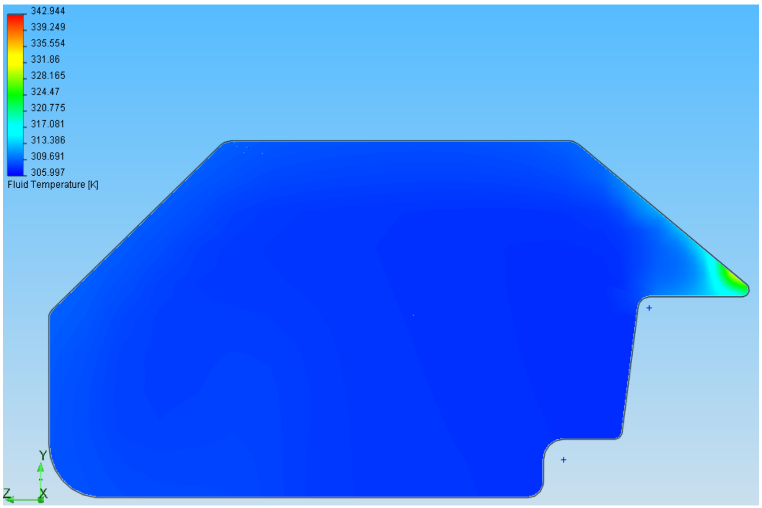


**Figure 4.19** Car cabin temperature distributions at 18000 seconds with heat extractor

Figure 4.19 shows the temperature distributions of cabin parked under direct sunlight after 18000 seconds with heat extractor. The simulation shows minimum temperature of 303K at most parts of the cabin and maximum temperature of 330K at the rear window. There is a small temperature difference at the rear window.

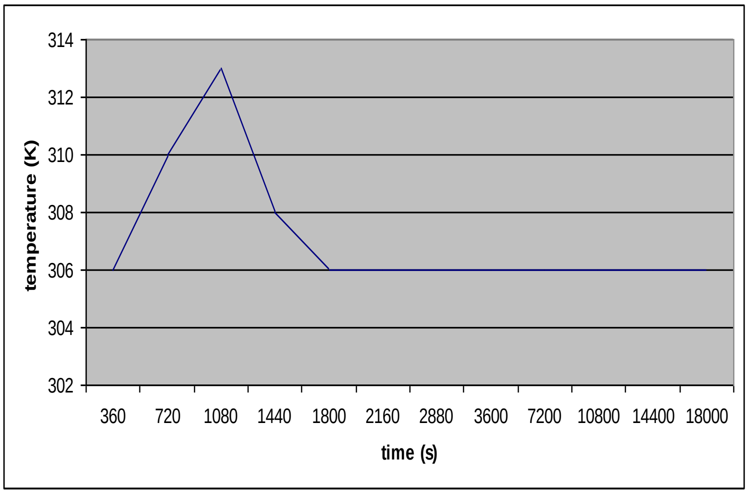


**Figure 4.20** Graph temperature versus time for with heat extractor

Graph shows that temperature rise is direct proportional to time. The temperature drops at 1800s to 306K as the fan starts. Then, temperature maintains at 306K because the fan circulates the air inside the cabin

### 4.2.16 1800 Seconds Tinted With Heat Extractor

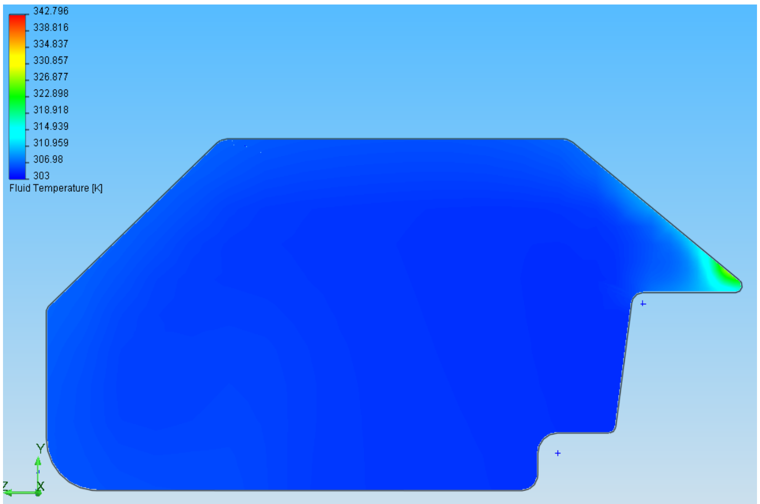


**Figure 4.21** Car cabin temperature distributions at 1800 seconds tinted with heat extractor

Figure 4.21 shows the temperature distributions of cabin parked under direct sunlight after 1800 seconds with heat extractor. The simulation shows minimum temperature of 303K at most parts of the cabin and maximum temperature of 330K at the rear window. There is a small temperature difference at the rear window.

### 4.2.17 3600 Seconds Tinted With Heat Extractor

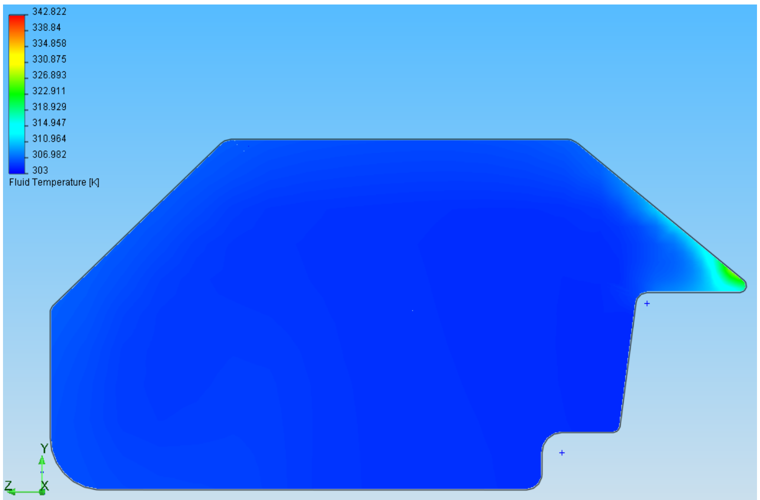


**Figure 4.22** Car cabin temperature distributions at 3600 seconds tinted with heat extractor

Figure 4.22 shows the temperature distributions of cabin parked under direct sunlight after 3600 seconds with heat extractor. The simulation shows minimum temperature of 303K at most parts of the cabin and maximum temperature of 330K at the rear window. There is a small temperature difference at the rear window.

### 4.2.18 18000 Seconds Tinted With Heat Extractor

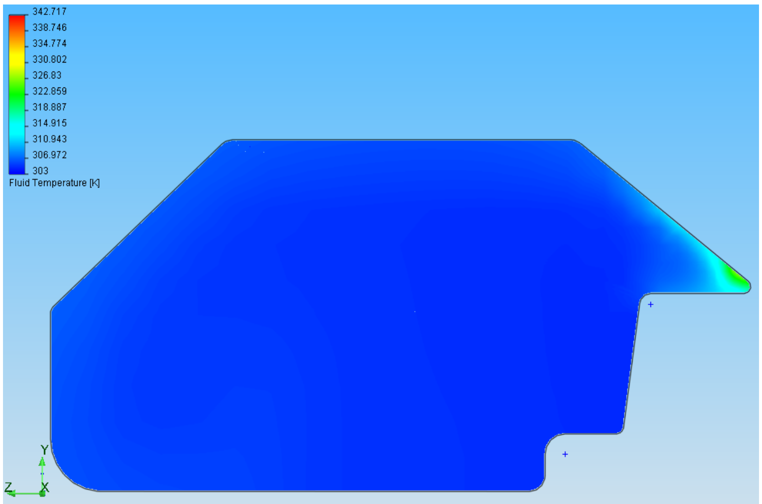


**Figure 4.23** Car cabin temperature distributions at 18000 seconds tinted with heat extractor

Figure 4.23 shows the temperature distributions of cabin parked under direct sunlight after 18000 seconds with heat extractor. The simulation shows minimum temperature of 303K at most parts of the cabin and maximum temperature of 330K at the rear window. There is a small temperature difference at the rear window.

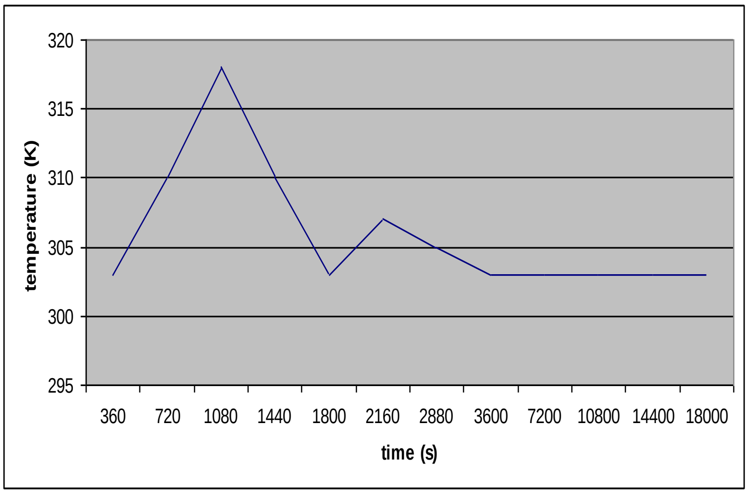


**Figure 4.24** Graph temperature versus time for tinted with heat extractor

Graph shows that temperature rise is direct proportional to time. The temperature drops at 1800s to 303K as the fan starts. Then, temperature increases up to 307K and drops again. Finally, maintains at 303K because the fan circulates the air inside the cabin.

### 4.2.19 1800 Seconds After Air Conditioning With Heat Extractor

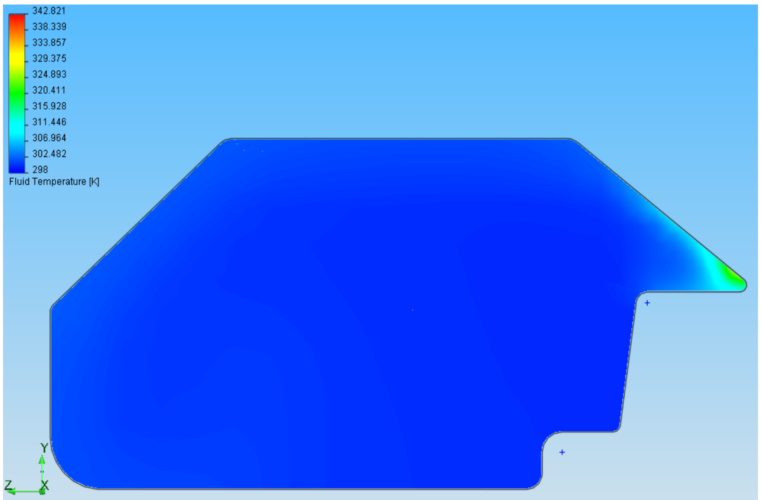


**Figure 4.25** Car cabin temperature distributions at 1800 seconds after air-conditioning with heat extractor

Figure 4.25 shows the temperature distributions of cabin parked under direct sunlight after 1800 seconds with heat extractor. The simulation shows minimum temperature of 298K at most parts of the cabin and maximum temperature of 325K at the rear window. There is a small temperature difference at the rear window.

### 4.2.20 3600 Seconds After Air Conditioning With Heat Extractor

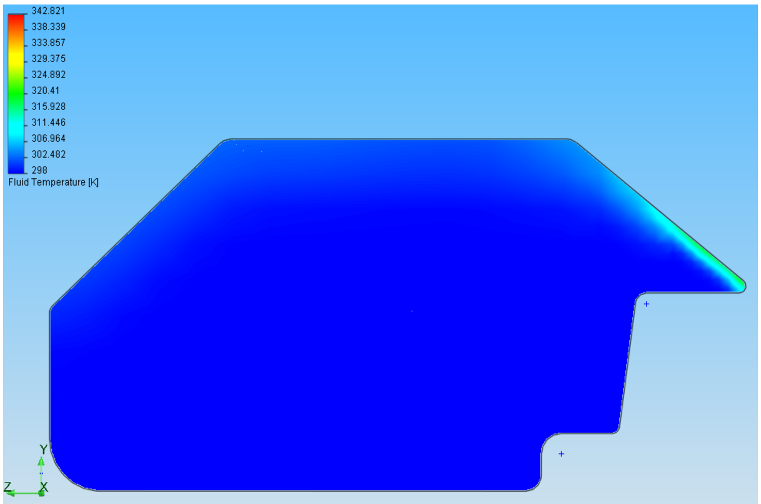


**Figure 4.26** Car cabin temperature distributions at 3600 seconds after air-conditioning with heat extractor

Figure 4.26 shows the temperature distributions of cabin parked under direct sunlight after 3600 seconds with heat extractor. The simulation shows minimum temperature of 298K at most parts of the cabin and maximum temperature of 316K at the rear window. There is a small temperature difference at the rear window.

### 4.2.21 18000 Seconds After Air Conditioning With Heat Extractor

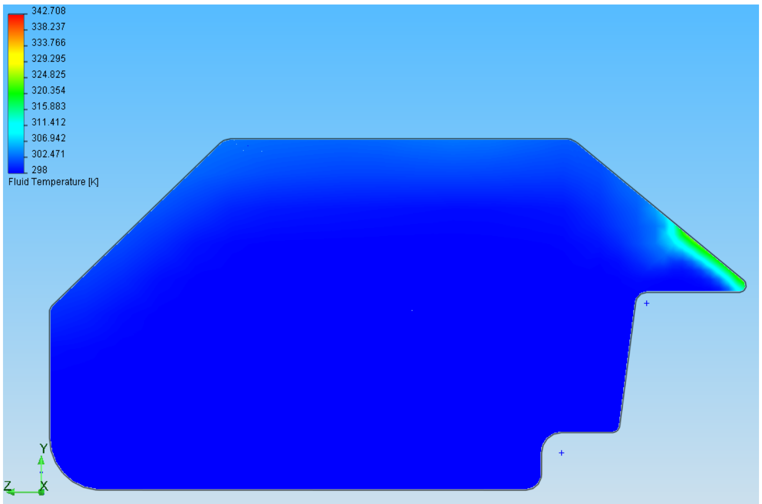

**Figure 4.27** Car cabin temperature distributions at 18000 seconds after air-conditioning with heat extractor

Figure 4.27 shows the temperature distributions of cabin parked under direct sunlight after 18000 seconds with heat extractor. The simulation shows minimum temperature of 298K at most parts of the cabin and maximum temperature of 325K at the rear window. There is a small temperature difference at the rear window.

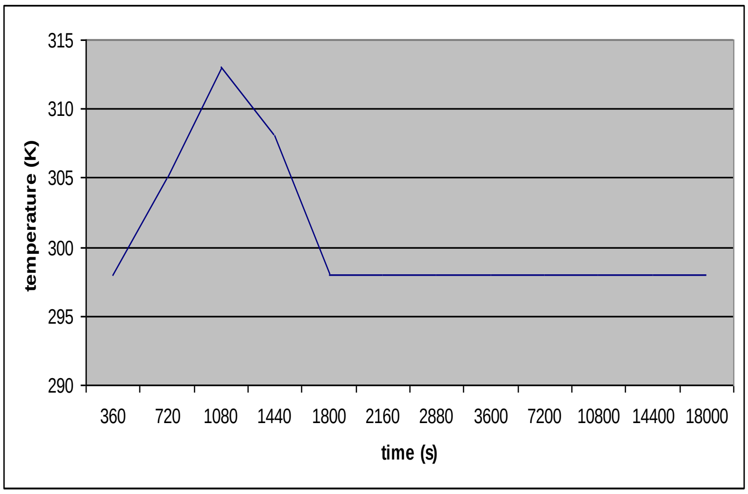


**Figure 4.28** Graph temperature versus time for tinted with heat extractor

Graph shows that temperature rise is direct proportional to time. The temperature drops at 1800s to 301K as the fan starts. Then, temperature maintains at 301K because the fan circulates the air inside the cabin.

### 4.2.22 1800 Seconds Tinted After Air Conditioning With Heat Extractor

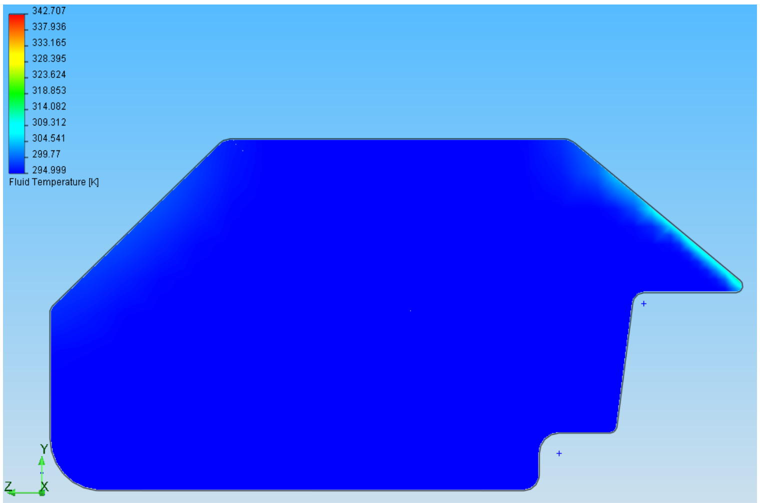


**Figure 4.29** Car cabin temperature distributions at 1800 seconds tinted after air-conditioning with heat extractor

The figure 4.29 shows the temperature distributions of cabin parked under direct sunlight after 1800 seconds with heat extractor. The simulation shows minimum temperature of 295K at most parts of the cabin and maximum temperature of 310K at the rear window. There is a small temperature difference at the rear window.

### 4.2.23 3600 Seconds Tinted After Air Conditioning With Heat Extractor

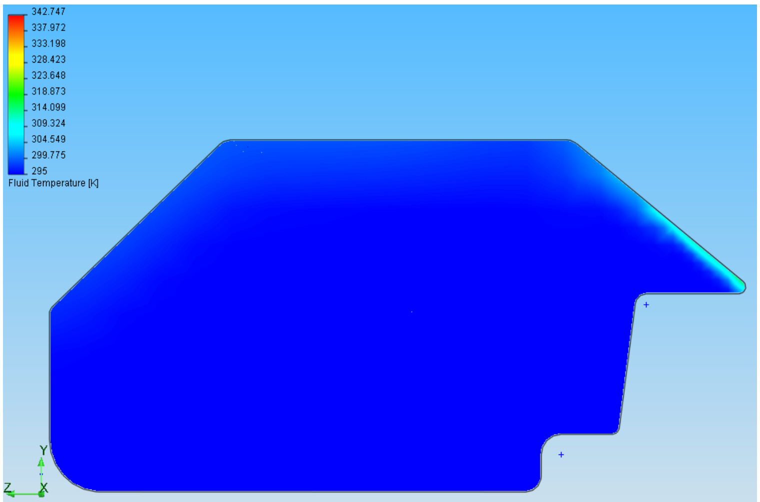


**Figure 4.30** Car cabin temperature distributions at 3600 seconds tinted after air-conditioning with heat extractor

Figure 4.30 shows the temperature distributions of cabin parked under direct sunlight after 3600 seconds with heat extractor. The simulation shows minimum temperature of 295K at most parts of the cabin and maximum temperature of 310K at the rear window. There is a small temperature difference at the rear window.

### 4.2.24 18000 Seconds Tinted After Air Conditioning With Heat Extractor

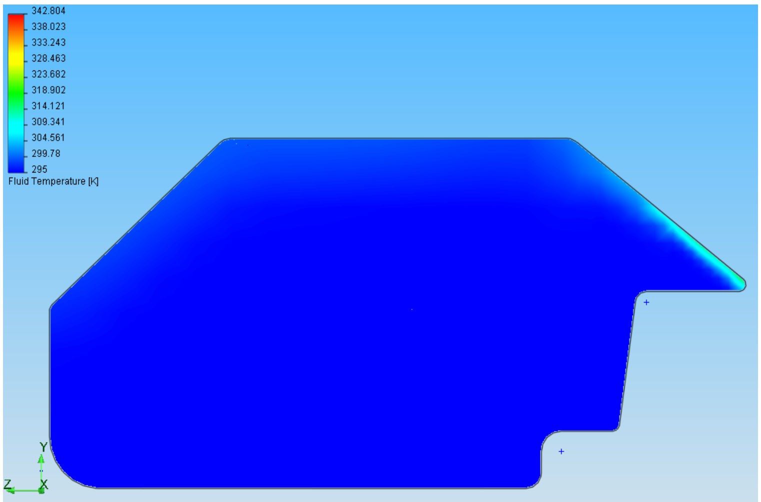


**Figure 4.31** Car cabin temperature distributions at 18000 seconds tinted after air-conditioning with heat extractor

Figure 4.31 shows the temperature distributions of cabin parked under direct sunlight after 18000 seconds with heat extractor. The simulation shows minimum temperature of 295K at most parts of the cabin and maximum temperature of 310K at the rear window. There is a small temperature difference at the rear window.

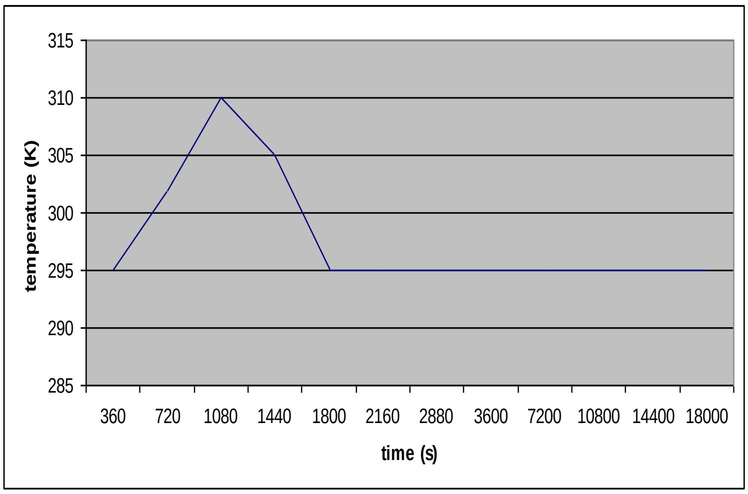

**Figure 4.32** Graph temperature versus time for tinted after air-conditioning with heat extractor

Graph shows that temperature rise is direct proportional to time. The temperature drops at 1800s to 295K as the fan starts. Then, temperature maintains at 295K because the fan circulates the air inside the cabin.

# CHAPTER 5

# CONCLUSION AND RECOMMENDATION

## 5.1 Conclusion

Reducing the cabin temperature of vehicle parked under direct sunlight has been one of the key issues in the invention process over the years. Reduced temperature results in improved passenger comfortness and lifetime of vehicle interior particularly accessories and leather parts. It is well known that significant difference is noticeable at rear window at all considerations and time. One of the most significant discoveries in the last decade in this area of research has been the finding of a heat extractor which could assist in reducing cabin temperature.

As the project is to prove that there is also an improvement by adding heat extractor device to the firewall of the vehicles which results decreased temperature variation and distribution in the cabin. This simulation using CFD also proved that the previous researcher, journalist and designer produced a necessary paper to design and analyze the heat extractor. This device not only maintains cabin temperature at optimum point but also reduces strain on air-conditioning system during start-up.

This project started from scratch where the measurement is taken from actual car model and by using a reference from other trusted resources the measurement transferred to 3-D modeling software. Then, the first step is to simulate the model using CFD – simple and widely used software to analyze the temperature and temperature distribution in cabin. All the data were taken as it will compared with the data after the heat extractor fixed. From the analysis, there is difference before and after the heat extractor is added. This difference can be seen clearly even by input data for both conditions.

For the beginning the objective and the scope of this thesis only focused on the temperature distribution in cabin. This is because room for improvement and recommendation still available because the thermal fluid field is very wide. Everything connected to temperature distribution still left with loads to discover. This project of the passenger vehicle is only a part of the gigantic world of thermal fluid analysis.

After all simulation and analysis completed, the objectives achieved, all the scopes successfully fulfilled.

## 5.2 Recommendation

For future researches, different flowrate and type of heat extractor can be compared by simply attached to vehicle to see the temperature distribution and cooling rate.

Some other recommendations;-

- Analysis using complete interior and accessories.
- Analysis using other CFD software such as Fluent.
- Designing electrical and electronic circuit for the heat extractor.
- Building duct for the heat extractor.
- Combining or connecting the heat extractor system to HVAC system.

# REFERENCES


1. Barton, Lyndon. 1993. *Mechanism Analysis: Simplified Graphical and Analytical Techniques 2nd ed*. New York: Marcel Dekker Inc.
2. Gilles, Tim. 2004. *Automotive Service, Inspection, Maintenance and Repair 2nd ed*. Thomson Delmar Learning.
3. Waldron, Kenneth and Kinzel, Gory. 1999. *Kinematics, Dynamics and Design of Machinery*. New York: John Wiley and Sons Inc.
4. A. Çengel, Yunus and A. Boles, Michael. 2007. *Thermodynamics: An Engineering Approach 6th ed.* Mc Graw Hill.
5. Finnemore, John and B.Franzini, Joseph. 2002. *Fluid Mechanics with Engineering Applications 10th ed.* Mc Graw Hill.
6. Ackers, P., W.R.White, A.J.Harrison and J.A.Perkins. 1978. *Weirs and Flames for Flow Measurement.* New York: Wiley Inc.
7. Eggleston, D.M. and F.S. Stoddard. 1987. *Wind Turbine Engineering Design.* New York: Van Nostrand Reinhold.
8. Peyret, R. and T.D. Taylor. 1990. *Computational Methods for Fluid Flow*. New York: Springer-Verloy.
9. Krieger, Malabar, FL. 1993. *Centrifugal and Axial Flow Pumps: Theory, Design and Application 2nd ed*. New York: Wiley Inc.
10. Wang, S. K. 2001. *Handbook of Air Conditioning and Refrigeration*.
11. Myszka, David H. 2005. *Machines & Mechanism : Applied Kinematic Analysis 3rd ed*. New Jersey: Pearson Education Inc.
12. Eck Bruno. 1973. *Fans: Design and Operation of Centrifugal, Axial-Flow and Cross Flow Fans*. Pergamon Press.
13. Taniguchi, Yousuke.. 2001. *R&D Review of Toyota CRDL Vol. 36 No. 2,* Kansei-Psychology Labs, Japan.

14. John P. Rugh, Lawrence Chaney, Jason Lustbader, John Meyer, Mukesh Rustagi, Kurt Olson, Rupert Kogler. 2006. *Reduction in Vehicle Temperatures and Fuel Use from Cabin Ventilation, Solar-Reflective Paint, and a New Solar-Reflective Glazin,* SAE International.

15. Komoriya T.: 1989. *Analysis of vehicle passenger compartment ventilation using experimental and numerical model*, *SAE paper No. 890312*.

16. Khan, M.U., Kawaguchi, K., Okui, Ohba, H. 2002. *Temperature Mitigation of Parked Automobile Under Blazing Sun.* 41st Heat Transfer Symposium.

# APPENDICES

## GANTT CHART FOR FINAL YEAR PROJECT 1

| Project Activities | W1 | W2 | W3 | W4 | W5 | W6 | W7 | W8 | W9 | W10 | W11 | W12 | W13 |
|---|---|---|---|---|---|---|---|---|---|---|---|---|---|
| **Project Topic Discussion**<br>~ research about title<br>~ Determination of project scope and objective | ■ | ■ | | | | | | | | | | | |
| **Literature Study**<br>~ Study on heat transfer<br>~ explore related software | | ■ | ■ | ■ | ■ | ■ | ■ | ■ | ■ | ■ | ■ | ■ | ■ |
| **Measurement**<br>~ Measure dimension of Proton Iswara Hatchback's Cabin | | | | ■ | ■ | ■ | ■ | ■ | | | | | |
| **Modeling**<br>~ Modeling Proton Iswara Hatchback's Cabin using 3D modeling software | | | | | | | ■ | ■ | ■ | ■ | ■ | | |
| **Simulation**<br>~ Simulation of hot air flow inside the cabin using CFD software | | | | | | | | ■ | ■ | ■ | ■ | ■ | ■ |

## GANTT CHART FOR FINAL YEAR PROJECT 2

| Project Activities | W1 | W2 | W3 | W4 | W5 | W6 | W7 | W8 | W9 | W10 | W11 | W12 | W13 |
|---|---|---|---|---|---|---|---|---|---|---|---|---|---|
| **Analysis**<br>~ Analysis on air flow characteristics inside the cabin | ■ | ■ | ■ | | | | | | | | | | |
| **Improvement / Modeling**<br>~ Choose location to fix fan<br>~ analysis on temperature and velocity of hot after adding the fan | | ■ | ■ | ■ | ■ | ■ | | | | | | | |
| **Evaluation**<br>~ collect all simulation data before and after the fan added | | | | | ■ | ■ | ■ | ■ | ■ | ■ | ■ | | |
| **Report writing**<br>~ Report preparation and submission | ■ | ■ | ■ | ■ | ■ | ■ | ■ | ■ | ■ | ■ | ■ | ■ | ■ |

## SUPERVISOR'S DECLARATION

We hereby declare that we have checked this project and in our opinion this project is satisfactory in terms of scope and quality for the award of the degree of Bachelor of Mechanical Engineering with Automotive

Name of Supervisor :
Position :
Date :

Name of Panel :
Position :
Date :

## STUDENT'S DECLARATION

I hereby declare that the work in this thesis is my own except for quotations and summaries which have been duly acknowledged. The thesis has not been accepted for any degree and is not concurently submitted for award of other degree.

Name :
ID Number :
Date :

## ACKNOWLEDGEMENT

I would like to express my deepest appreciation and gratitude to my supervisor, Mr. Devarajan Ramasamy for his guidance, patience for giving advises and support throughout the progress of this project. A very special thanks to all lecturers and vocational trainers for the guidance, experience sharing and comment on my project thesis. They were not hesitant to answer all my doubts and spending their time to guide me during my experimental work.

A great appreciation is acknowledged to the Faculty of Mechanical Engineering for the funding under the final year project.

Last but not least, I would like to thank all my friends for their support and encouragement given to me, especially during the hard times.

## ABSTRACT

A vehicle parked under direct sunlight for a few hours will have a hot cabin. Besides creating an uncomfortable situation, this actually affects the lifetime of the vehicle's interior and accessories such as seats, carpets and door panels. We experience it all the time when we park under the hot sun or in a non-cooled garage. After only a short time our car becomes a virtual oven. We open the door and the oppressive heat hits us. Everything we touch burns like a hot plate. Our dashboard vinyl eventually begins to crack and our leather interior becomes dry and brittle. The health risk involved when entering a sweltering car can damage our lungs and become life threatening. This happens because air flow from interior to exterior takes place in a very small volume and velocity. The simulation started by taking temperature, flow velocity, type of flow, cabin volume and time into consideration. A Proton Iswara Hatchback's cabin used as the air flow boundary. The simulation shows difference in air flow and temperature out of the cabin when a couple of DC fans installed to the spots where the most heat is generated. The expected result is that the increase in air flow from cabin to out of the car would decrease the temperature in the cabin. In this project, the main idea is to increase the air flow out of the car as much as possible to ensure the cabin temperature is kept to the level intended. As the conclusion, the added fans give impact to the cabin temperature of car.

# ABSTRAK

Sesuatu kenderaan yang diletakkan dibawah sinaran matahari secara langsung akan mempunyai bahagian dalam yang panas. Di samping menyumbang kepada keadaan yang kurang selesa, hal ini juga sebenarnya mengurangkan jangka hayat bahagian dalam dan aksesori seperti tempat duduk, karpet serta panel pintu. Kita menghadapi situasi sebegini setiap masa kereta diletakkan di bawah sinaran matahari terik ataupun di tempat yang kurang pengudaraan. Pada masa yang singkat, kereta menjadi panas seperti ketuhar. Apabila kita membuka pintu, udara panas yang kurang menyenangkan kena pada kita. Setiap apa yang kita pegang adalah seperti pinggan panas. Vinyl dashboard retak dan bahagian dalam kulit menjadi kering dan rapuh. Memasuki kereta sebegini memberi kesan buruk terhadap kesihatan kita. Keadaan ini adalah disebabkan oleh pemindahan udara panas dari dalam kenderaan ke luar berlaku pada tahap yang amat rendah. Simulasi dimulakan dengan mengambilkira suhu, halaju aliran haba, jenis aliran haba, isipadu bahagian dalam kenderaan dan masa. Bahagian dalam Proton Iswara Aeroback dijadikan bahan ujikaji. Simulasi menunjukkan perbezaan dalam pemindahan haba dan suhu dari dalam kereta apabila beberapa buah kipas dipasang di dalam kereta pada tempat-tempat tertentu yang dikenalpasti sebagai tempat suhu yang tinggi. Keputusan yang dijangka adalah apabila kadar pemindahan haba yang tinggi berlaku dari dalam kenderaan ke luar, maka kadar suhu bahagian dalam kenderaan akan turun. Projek ini bermatlamat untuk meninggikan kadar pemindahan haba ke luar kenderaan supaya suhu dalam kenderaan dikekalkan pada tahap tertentu. Sebagai kesimpulan, pemasangan kipas pada tempat-tempat tertentu memberi impak kepada suhu bahagian dalam kereta.

# TABLE OF CONTENTS

# LIST OF TABLES

# LIST OF FIGURES

## LIST OF SYMBOLS

| | |
|---|---|
| $\rho$ | Density |
| $\omega$ | Angular Velocity |
| V | Velocity |
| M | Mass Flowrate |
| r | Radius |
| P | Pressure |
| T | Temperature |
| A | Area |
| R | Gas Constant |
| $\pi$ | Pie |

# LIST OF ABBREVIATIONS

| | |
|---|---|
| CAD | Computer Aided Design |
| CFD | Computational Fluid Dynamics |
| 2-D | Two Dimensional |
| 3-D | Three Dimensional |
| HEV | Hybrid Electric Vehicle |
| HVAC | Heating, Ventilating and Air-conditioning |
| ANSI | American National Standards Institute |
| ISO | International Organization for Standardization |
| DIN | German Institute for Standardization |
| GOST | Euro-Asian Council for Standardization |
| JIS | Japanese Standards Association |
| BSI | British Standards |
| GB | Guojia Biaozhun, Chinese national standard |
| m | meter (length) |
| RPM | Revolution per Minute |
| s | second(time) |
| kg | Kilogram |
| kJ | kilo joule |
| K | Kelvin |
| kPa | kilo Pascal |
| hr | hour |
| rad | radians |