Palestine Polytechnic University College of Engineering

Passive Natural Ventilation in Vehicles

By:

Sohaib Iseed Ahmad Iseed

Supervisor :

Dr.Hussein Amro.

Submitted to the College of Engineering in partial fulfillment of the requirements for the Bachelor degree in Mechanical Engineering

Palestine Polytechnic University

Hebron, December 2018

Palestine Polytechnic University

Collage of Engineering

Mechanical Engineering Department

Hebron – Palestine

Passive Natural Ventilation in Vehicles

Project Team:

Ahmad "Mohamad Taysir" Iseed

Sohaib "Mohamad Taysir" Iseed

Submitted to the Collage of Engineering

In partial fulfillment of the requirements for the

Bachelor degree in Mechatronics Engineering.

Supervisor Signature

....

Testing Committee Signature

....

Head of the Department Signature

....

2018/2019

Abstract

Temperature is the environmental factor that affect the human comfort level, passenger thermal comfort is mainly influenced by inside cabin of the vehicle temperature. In hot weather, when the vehicle parks in direct sunlight conditions, the cabin will form a high-temperature thermal environment and pose a serious threat for children and pets left in the car. The solutions proposed in the literature to reduce the temperature of the cabin during parking are depending on active systems, in other words taking advantage of the vehicle battery which may consume the battery power for long parking periods. Also, these solutions may use solar energy as an electrical source, but the safety is not granted because, these systems need vehicle window opening. A system working by a natural ventilation that passively reduce the temperature inside the vehicle when it is parked was made, the system reduced more than 20 C°. The passive ventilation system consists of decorated safe holes above and under the cabin of vehicle in order to make an air circulation between the outside and the inside of the vehicle due to the temperature gradient between inside the cabin and the temperature under the vehicle (shadow). Computational Fluid Flow (CFD) simulation is carried out by using ANSYS software in order to have the optimal design of the proposed system and the accurate results.

الملخص

درجة الحرارة هي العامل البيئي الذي يؤثر على مستوى راحة الإنسان ، وتتأثر راحة الركاب بشكل رئيسي بدرجة حرارة المقصورة الداخلية في الطقس الحار ، عندما تركن المركبة تحت اشعة الشمس المباشرة ، ستتشكل المقصورة بيئة حرارية عالية الحرارة وتشكل تهديدًا خطيرًا للأطفال والحيوانات الأليفة. تعتمد الحلول المقترحة في الدراسات السابقة لتقليل درجة حرارة الكابينة أثناء الوقوف على الأنظمة النشطة، بمعنى آخر الاستفادة من بطارية السيارة التي قد تستهلك طاقة البطارية لفترات الانتظار الطويلة. أيضا ، قد تستخدم هذه الحلول الطاقة الشمسية كمصادر كهربائية ، ولكن لا يتم منح السلامة لأن هذه الأنظمة تحتاج إلى فتح نافذة السيارة. ولذلك ، فإن الغرض من هذا المشروع هو تصميم نظام تهوية طبيعي يقلل من درجة الحرارة داخل المركبة مقصورة المركبة من حيث يتكون النظام من ثقوب آمنة مزينة فوق وتحت;عندما تكون متوقفة أجل إجراء تدوير الهواء بين الخارج وداخل السيارة بسبب الفرق في درجة الحرارة بين داخل باستخدام برنامج (CFX)المقصورة ودرجة الحرارة تحت المركبة (الظل). يتم تنفيذ محاكاة للنظام المقترح والنتائج الدقيقة.والغرض من مشروعنا هو إنشاء نظام يعمل عن طريق ANSYS التهوية الطبيعية دون استخدام أي طرق كهربائية حيث سوف يتم تصميم النظام عن طريق عمل .محاكاة لمقصورة المركبة

الاهداء

نهدي مشروعنا هذا إلى قدوتنا ومعلمنا وقائد أمتنا محمد صلى اهلل عليه وسلم

إلى من هم أكرم منا مكانة .. شهداء فلسطين

إلى من ضحوا بحريتهم من أجل حرية غيرهم ..الاسرى والمعتقلين

إلى كل من ساهم في وصولنا لطريق النهاية .. إلى كل من علمنا شيئاً جديدا

وغذى فكرنا بالعلم والمعرفة .. إلى كل من وقف بجانبنا وساعدنا في كل المصاعب

إلى دكاترتنا وأساتذتنا في الجامعة

إلى من رعونا بنور قلبهم .. وحمونا بحكمتهم .. وقدموا لنا حنانهم وقلبهم .. إلى من سقونا وأطعمونا وربونا وأدبونا ومنحونا الحب والحنان .. إلى من فرشوا طريقنا بالورود .. ورافقونا في الصعود .. إلى من فتحوا أمامنا أبواب التفوق و النهوض وكسر قيود الظالم .. إلى بسمة الامل ونبع الحنان .. إلى من هم بلسم روحنا .. إلى ورود حياتنا إلى من رعونا بحنانهم .. إليكم والدينا

إلى من يحملون في عيونهم ذكريات طفولتنا وشبابنا .. إخوتنا وأخواتنا

إلى من كانو ملاذنا وملجئنا .. إلى من تذوقنا معهم أجمل اللحظات .. إلى من سنفتقدهم وأتمنى أن بفتقدونا

إلى من جعلهم اهلل إخوتنا باالله .. ومن أحببناهم باالله أصدقائنا في الجامعة

إهدائنا إلى من جمع بين سعادتنا وحزننا .. إلى من لم نعرفهم .. ولم

يعرفونا إلى من نتمنى أن نذكرهم .. إذا ذكرونا .. إلى من نذكرهم ونسونا

Contents

List of figures.

1

Chapter One

Introduction

1.1 Introduction

1.2 problem Definition

1.3 project Objective

1.4 Task Table

1.5 Time tables

1.1 Introduction

As the summer approaches, the problem of rising temperatures in the vehicles appeared when exposed directly to the sunlight for a long time, with no ventilation and air conditioning takes place so that vehicles sometimes turn out to a real sauna rooms in a short time, and increase the temperature to high number. Many vehicles owner seek to park their vehicles under trees or covered areas. Even turning on the air conditioner in the first few minutes does not reduce the temperature efficiently and quickly as shown in fig 1.1 below.

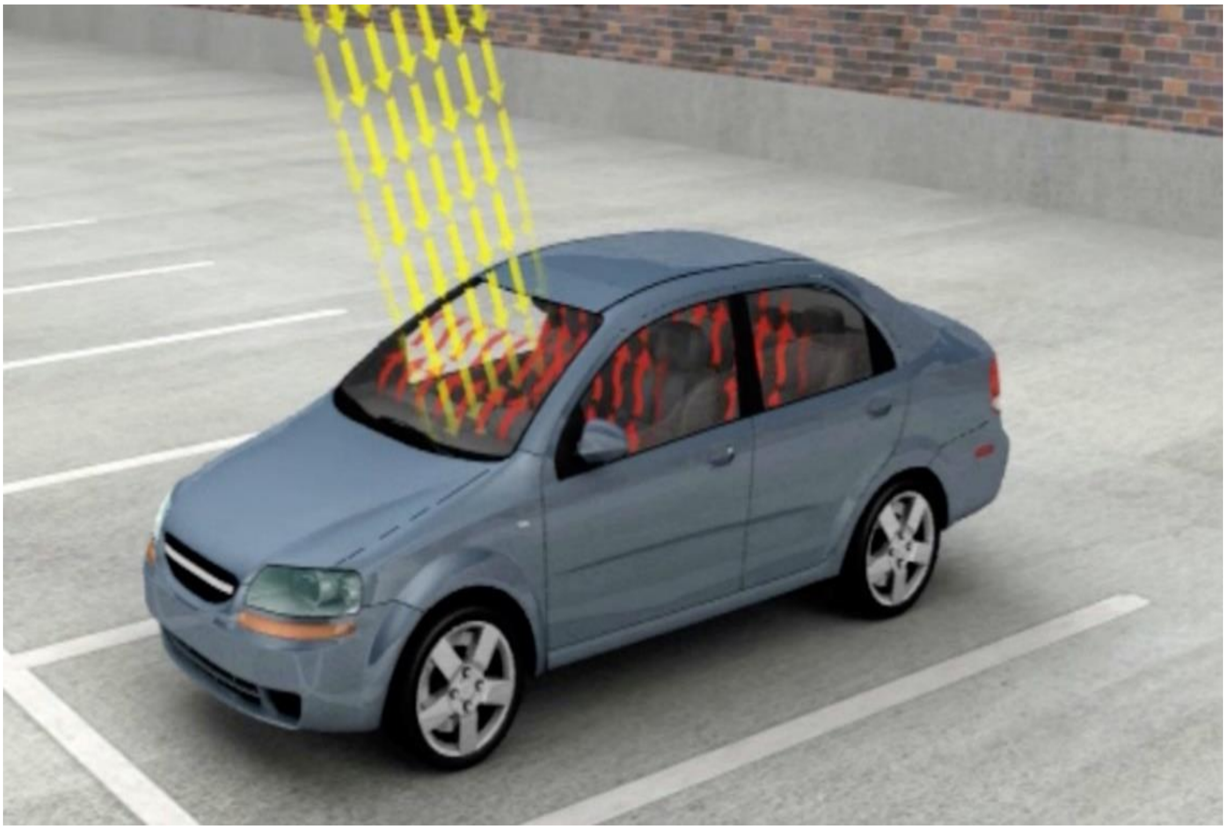


Figure 1.1 Rising Temperatures in the Vehicle.

The amount of Oxygen (O_2) that existed inside a closed vehicles is less than the Oxygen outside , when the vehicle locked which means we close the doors and the windows, this for sure will prevents the fresh air to enter the cabin of the vehicle and this will increase the concentration of Carbon dioxide (CO_2) and reduce the amount of Oxygen (O_2).[1].

In a study measure the temperature inside cabin of vehicle, the vehicle was parked under the direct sunlight without any protection or roof for 3 hours. At 14:00 the cabin temperature reached 49.5 ºC, with the difference of 13.0 ºC compared to the outside / outdoor temperature (36.5 ºC). [1]

In the last time, there have been many cases of children death inside the vehicles in case of forgotten them alone by mistake, and this leads to suffocation, or exposure to heat stroke, due to the high temperature inside the cabin of the vehicle, the sunlight bound to transform the car into a sauna room in a few minutes as shown in fig1.2. Therefore, it was necessary to find a solution to solve the suffocation problems of children in the car.

Figure 1.2: Suffocation chilled in vehicle

Leaving the windows, open slightly does not prevent the temperature from rising to a dangerous level as show in in fig 1.3.

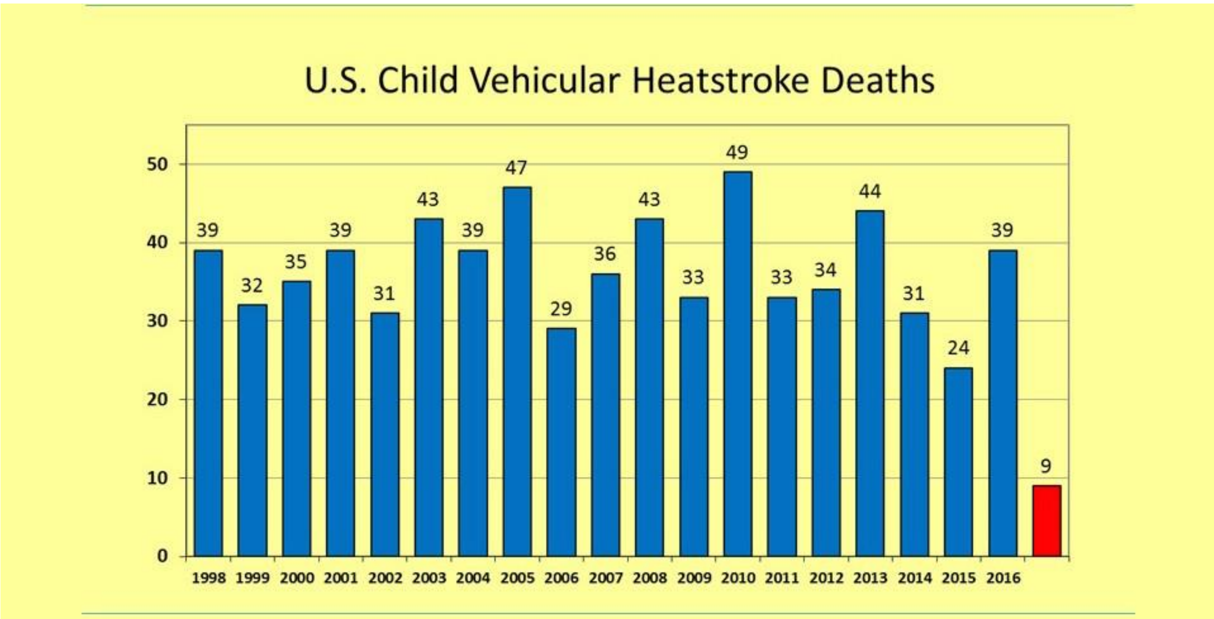


Figure 1.3: Child Vehicular Heatstroke Death

1.2 Problem Definition

The high increasing of temperature inside the cabin of the vehicle when its parked especially at the summer time that makes many problems and sides effect for driver also, increase the carbon dioxide and makes bad effects for the driver and passengers, therefore there must be an effective way to reduce and overcome this undesirable phenomenon.

1.2 Project Objectives

- Reducing the amount of high temperature inside the vehicles when it's parked under sunlight for long period so that the driver be able to drive the car comfortably.
- To save the life of children in case of leaving them inside vehicles and protecting them from the danger of getting suffocated.
- To reduces the amount of Carbon dioxide inside the car.

The main objective of this project is to overcome this phenomenon by creating and designing a passive natural ventilation system in vehicles to reduce the temperature inside the cabin by exploitation the temperature gradient inside cabin and the shadow's temperature in order to make a natural circulation for air when vehicle parked as shown in fig 1.4.

Figure 1.4: Vehicle's Shadow

1.3 Previous Studies

Palestine Polytechnic University project named Design and implementation a security system to protect children in the car from suffocation by ventilation system and utilize on solar energy. The project design a safety system to protect children from being suffocated inside vehicles through new ventilation system works on solar power, the system also serves to cool the vehicle while it is parked under sunlight before driving.

Palestine polytechnic university a project named smart automobile air conditioning and ventilation system that when the temperature inside the vehicle increased at certain level and when children forgetting in car the windows completely and automatically open by sensors to reduce the temperature inside the cabin and to save the life of the children [2].

Another attempt was also made to solve this problem by designing a device placed on windows of the vehicle called (Kuacar) this device running on solar energy, consist fan that expelled the air from inside to the outside to reduce the degree of temperature as shown in the fig 1.5.

Figure 1.5: Kulcar Device

Simulation and Analysis on Heat Transfer and Pre-cooling Characteristics of New Solar Power Vehicle Parking Ventilation System. It designed to reduce the temperature of the cab when the vehicle is parked by solar panel mounted on the roof of the vehicle to discharge heat inside the cabin[3]

Pushing-Pulling Based Vehicle Parking Ventilation Cooling.

In this system, temperature drops owing to fresh air pushed in by blower, while lower space temperature is mainly affected by the upper space air mixed with new fresh air, the system works on solar energy [4].

All these suggested solutions above were created to reduce the amount of high heat inside the cabin of the vehicles when its parked, but all of them were depending on solar energy and electrical sources, and this kind of methods have many disadvantage such as the energy will be eliminated by time and needs maintenance, it is easy to be stolen by anyone, but our project completely natural which not depend on any kind of electrical or energy sources.

1.4 Task Table

Table 1.1 :Task table

Task ID	Task Description
T1	Project selection
T2	Collection information for project
T3	Literature review
T4	Chapter one
T5	Simulation solid works
T6	Learning ANSYS software
T7	Do some variations if required
T8	Simulation solid works
T9	Simulation on ANSYS
T10	Test the result
T11	Writing the text
T12	Make a final adjustments on the text

1.5 Time Tables

Table 2.2 :Time table

	1st semester															
Task/Week	1	2	3	4	5	6	7	8	9	10	11	12	13	14	15	16
Project selection																
Collection information																
Literature review																
Chapter one																
Simulation on solid works																
Learning ANSYS software																

	Second semester															
Task/Week	1	2	3	4	5	6	7	8	9	10	11	12	13	14	15	16
Do some variations if required																
Simulation solid works																
Simulation on ANSYS																
Test the result																
Writing the text																
Make a final adjustments																

2

Chapter Two

Ventilation and heat transfer

2.1 Introduction

2.2 why Natural ventilation In Building

1.3 Heat Transfer Mechanisms

2.1 Introduction

A Natural Ventilation study can be used to optimize the thermal comfort and ensure adequate air quality is achieved within a building or room. Furthermore, adequately designed natural ventilation for a building enables the reduction in reliance on HVAC systems, hence reducing the buildings energy consumption. Different configurations of openings in a building are investigated to maximize wind draught and at the same time ensure a comfortable thermal environment as shown in figure2.1 below.

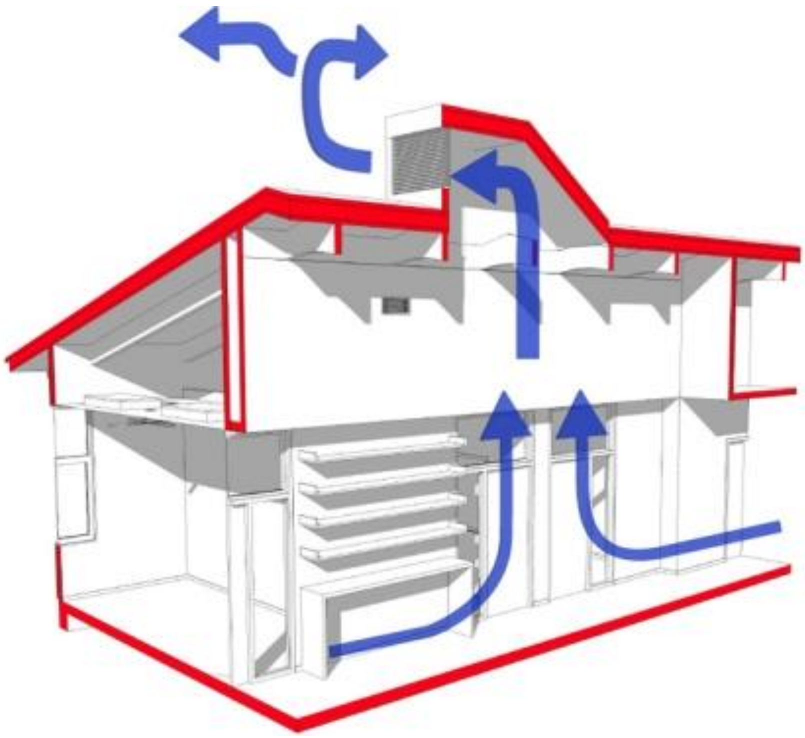
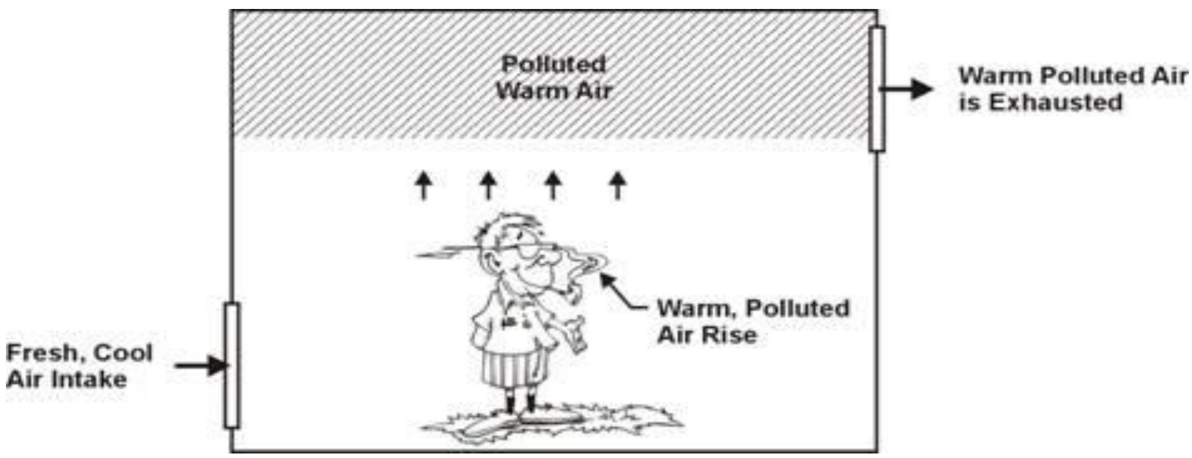


Figure 2.1: Natural Ventilation in Building

2.2 Why Natural Ventilation in Building.

Change the used air witch saturated with carbon dioxide, with fresh external air coming from outside and minimum ventilation for this purpose we need each room or space busy to expel the exhaust air and carbon dioxide resulting from the breathing process and supply us with oxygen show in fig 2.2.[5]

The removal of smoke occupied buildings require a high rate of ventilation to expel smells and smoke. Ventilation is also necessary to flush out combustion products from cooking in kitchens and to remove the wet vapors from cooking to prevent their deposition and condensation inside buildings. Cigarette smoking also increases the urgent need for Ventilation. These ventilation requirements are called hygienic ventilation[5]



/Figure 2.2: Removal of Smoke Occupied

Supplying clean healthy air and cooling the buildings from the inside with load currents. This comes when there is a difference in temperature between the inside and then inside.

This difference in temperature depends on how the temperature changes during the day (the heat obtained from the outside and the heat generated at home or the combination of all these factors and this type of ventilation is called structural cooling or structural[5].

2.3 Heat Transfer

Heat is defined in physics as the transfer of thermal energy across a well-defined boundary around a thermodynamic system; the thermodynamic free energy is the amount of work that a thermodynamic system can perform. Enthalpy is a thermodynamic potential, designated by the letter "H" that is the sum of the internal energy of the system (U) plus the product of pressure (P) and volume (V). Joule is a unit to quantify energy, work, or the amount of heat .

Heat transfer is a process function (or path function), as opposed to functions of state, the amount of heat transferred in a thermodynamic process that changes the state of a system depends on how that process occurs, not only the net difference between the initial and final states of the process [6].

Heat Transfer Mechanisms 2.4

Heat transfer mechanisms can be grouped into three broad categories

Conduction, Conviction and Radiation as shown in fig2.3.

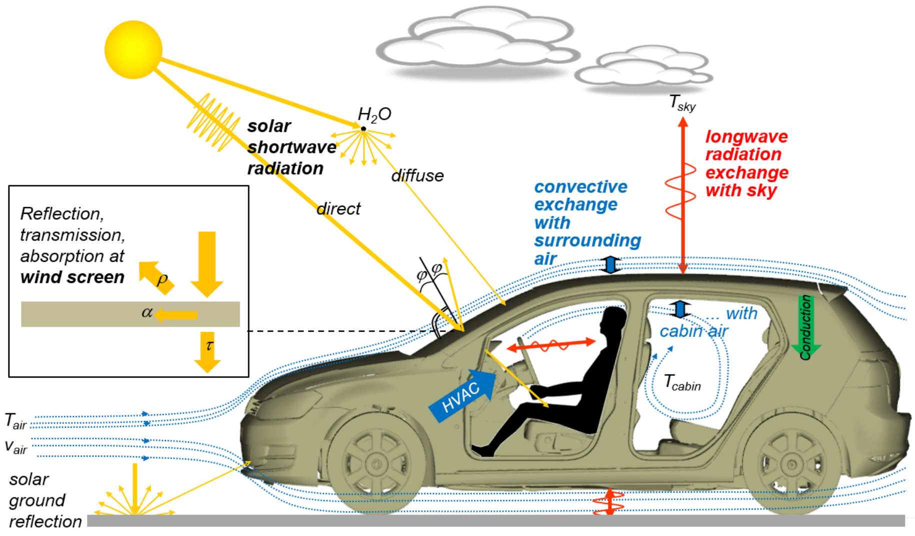


Figure 2.3: Heat Transfer Mechanisms

Convection

Convective heat transfer, often referred to simply as convection, is the transfer of heat from one place to another by the movement of fluids and the ambient load is the contribution of the thermal load transferred to the cabin air due to temperature difference between the ambient and cabin air. In the project type of convection, Convection free is a result of exchange between the air inside the car and the outside air [6].

Radiation

Radiation heat transfer is an energy transport due to emission of electromagnetic waves or photons from a surface or volume. The radiation does not require a heat transfer medium, and can occur in a vacuum, in our project the heat is transfer from the sun to the car through the front and back windows [6] .

2.5 Measurements

After measuring the temperature in a indifferent positions for vehicle parked five hours in Wad Al Hareya area, the temperature in front position was 65^0 C as showing in fig 2.4 , measured temperature in upper position was 56.4^0 C as shown in fig 2.5 , the measured temperature in the back of the vehicle was 53^0 C shown in fig 2.6, measured temperature under the vehicle which is the temperature of the shadow was 23.8^0 C as shown in fig 2.7.

Figure 2.4: Measured Temperature in Vehicle Front position

Figure 2.5: Measured Temperature in Vehicle Above position

Figure 2.6: Measured Temperature in Vehicle Back position

/**Figure 2.7:** Measured Temperature Under Vehicle

3

Chapter Three

Natural convection under ANSYS

3.1 Introduction

Natural convection is a mechanism, or type of heat transport, in which the fluid motion is not generated by any external source (like a pump, fan, suction device, etc.) but only by density differences in the fluid occurring due to temperature gradients. In natural convection, fluid surrounding a heat source receives heat and by thermal expansion becomes less dense and rises. The surrounding, cooler fluid then moves to replace it. This cooler fluid is then heated and the process continues, forming a convection current; this process transfers heat energy from the bottom of the convection cell to top.

ANSYS is a general purpose software, used to simulate interactions of all disciplines of physics, structural, vibration, fluid dynamics, heat transfer and electromagnetic for engineers.

So ANSYS, which enables to simulate tests or working conditions, enables to test in virtual environment before manufacturing prototypes of products. Furthermore, determining and improving weak points, computing life and foreseeing probable problems are possible by 3D simulations in virtual environment.[7] .

ANSYS software with its modular structure gives an opportunity for taking only needed features. ANSYS can work integrated with other used engineering software on desktop by adding CAD and FEA connection modules.

ANSYS can import CAD data and also enables to build a geometry with its "preprocessing" abilities. Similarly, in the same preprocessor, finite element model (a.k.a. mesh) which is required for computation is generated. After defining loadings and carrying out analyses, results can be viewed as numerical and graphical [7].

ANSYS Workbench is a platform, which integrate simulation technologies and parametric CAD systems with unique automation and performance. The power of ANSYS Workbench comes from ANSYS solver algorithms with years of experience. Furthermore, the object of ANSYS Workbench is verification and improving of the product in virtual environment as shown in fig 3.

ANSYS Workbench, which is written for high-level compatibility with especially PC, is more than an interface and anybody who has an ANSYS license can work with ANSYS Workbench. As same as ANSYS interface, capacities of ANSYS Workbench are limited due to possessed license.

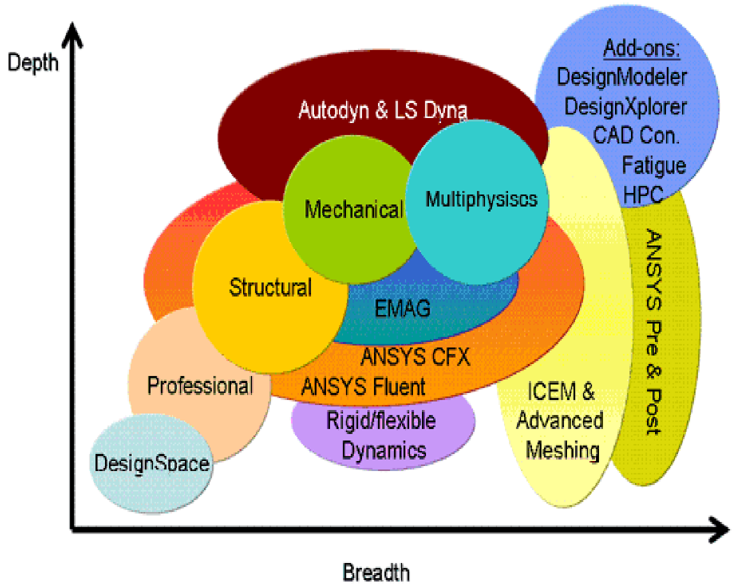


Figure 3.1: Disciplines of ANSYS

3.2 Natural convection under ANSYS depends on many numbers, equations and Algorithms such as

3.2.1Types of Numbers used

3.2.1.1 Reynolds number:

The Reynolds number is the ratio of inertial forces to viscous forces within a fluid which is subjected to relative internal movement due to different fluid velocities, in which is known as a boundary layer in the case of a bounding surface such as the interior of a pipe[8].

The Reynolds number (Re) is defined as:

$$\mathrm{Re} = \frac{\rho U L}{\mu} \qquad (3.1)$$

Where ρ the density of the fluid is, U is the velocity of the fluid (m/s), L is a characteristic linear dimension (m), and μ is the dynamic viscosity of the fluid.

And may be interpreted as a measure of the relative importance of advection (inertia) to diffusion (viscous) momentum fluxes. If the momentum fluxes are in the same direction then the Reynolds number reveals the boundary layer characteristics of the flow. If the fluxes are defined such that the diffusion is in the cross stream direction, then as shown in Fig. 3.2 Re conveys the flow regime (i.e. laminar, transitional, or turbulent)[8].

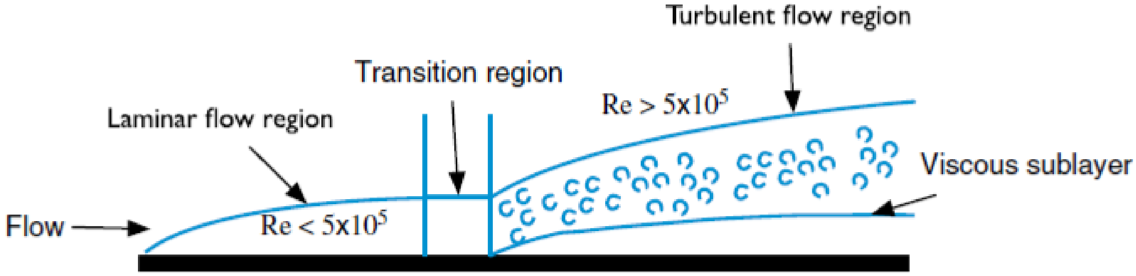


Figure 3.2 laminar, transitional, or turbulent

An example showing the flow field for different values of Reynolds number is depicted in Fig. 3.3.

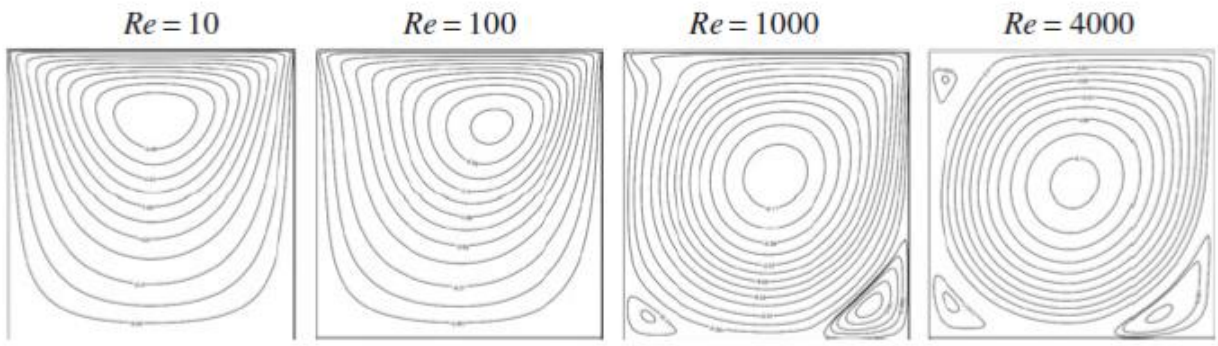


Figure 3.3 flow field for different values of Reynolds number

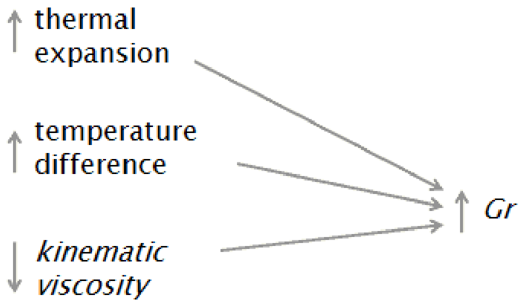
3.2.1.2 Grashof Number

The Grashof number is a dimensionless number, named after Franz Grashof. ,is defined as the ratio of the buoyant to viscous force acting on a fluid in the velocity boundary layer. Its role in natural convection is much the same as that of the Reynolds number in forced convection. [9].

Natural convection is used if this motion and mixing is caused by density variations resulting from temperature differences within the fluid. Usually the density decreases due to an increase in temperature and causes the fluid to rise. The buoyant force causes this motion. The major force that resists the motion is the viscous force. The Grashof number is a way to quantify the opposing forces. [9].

The Grashof number is defined as:

$$Gr = \frac{\text{buoyant forces}}{\text{viscous forces}} = \frac{g\beta(T_{wall} - T_{\infty})L^3}{\nu^2} \quad \text{.......(3.2)}$$



where:

g = gravitational acceleration, m/s2

β = coefficient of volume expansion, 1/K (β= 1/T for ideal gases)

Ts = temperature of the surface, ˚C

T_∞ = temperature of the fluid sufficiently far from the surface, ˚C

Lc = characteristic length of the geometry, m

ν= kinematic viscosity of the fluid, m2 /s.

3.2.1.3 Nusselt number (Nu) .

Nusselt number it is a dimensionless number, and it is the ratio of convection to fluid conduction heat transfer across (normal to) the boundary ,The convection and conduction heat flows are parallel to each other and to the surface normal of the boundary surface, and are all perpendicular to the mean fluid flow in the simple case. [9].

$$\overline{Nu}_L = \frac{\bar{h}L}{k} \quad \text{...........(3.3)}$$

where h is the convective heat transfer coefficient of the flow, L is the characteristic length, k is the thermal conductivity of the fluid.

.2.2 Types of Equations used 3

3.2.2.1 Conservation of mass (continuity equation):

The principle of conservation of mass indicates that in the absence of mass sources and sinks, a region will conserve its mass on a local level[8].

Derivation of continuity equation:

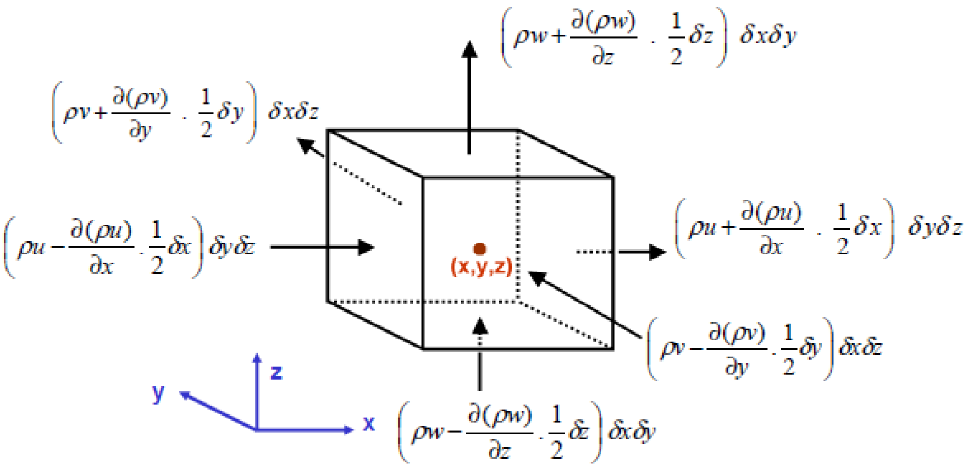


Figure 3.4 fluid element

The Change in the mass of the system equals the change in the control volume plus the mass in and mass out:

For a steady flow:

$$0=\frac{\partial \rho u}{\partial x}+\frac{\partial \rho v}{\partial y}+\frac{\partial \rho w}{\partial z} \qquad (3.4)$$

For incompressible flow:

$$0 = \frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z} \qquad (3.5)$$

For two dimension flow:

$$0 = \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z} \qquad (3.6)$$

3.2.2.2 Conservation of linear momentum:

The principle of conservation of mass indicates that in the absence of any external force acting on a body .the body retains its total momentum.

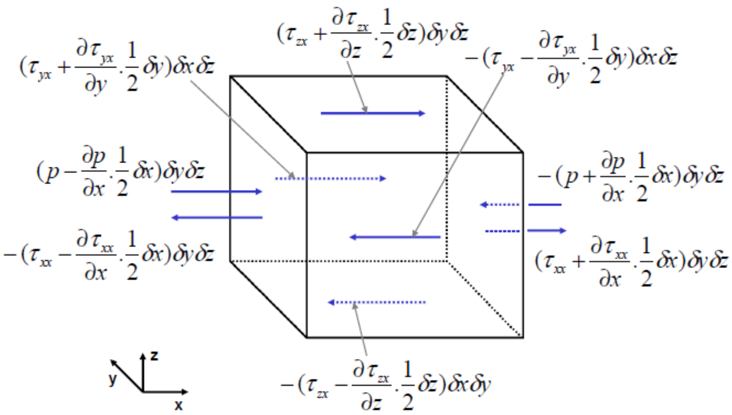


Figure 3.5 Infinitesimally small, moving fluid element.

Only the forces in the x direction are shown

The equation in the x-direction:

$$\left(\frac{\delta \sigma_{xx}}{\delta x} + \frac{\delta \tau_{yx}}{\delta y} + \frac{\delta \tau_{zx}}{\delta z}\right) + \rho g_x = \rho\left(\frac{\partial u}{\partial t} + u\frac{\partial u}{\partial x} + v\frac{\partial u}{\partial y} + w\frac{\partial u}{\partial z}\right)$$

(3.7)

The equation in the y-direction:

$$\left(\frac{\delta\sigma_{yy}}{\delta y}+\frac{\delta\tau_{xy}}{\delta x}+\frac{\delta\tau_{zy}}{\delta z}\right)+\rho g_y=\rho\left(\frac{\partial v}{\partial t}+u\frac{\partial v}{\partial x}+v\frac{\partial v}{\partial y}+w\frac{\partial v}{\partial z}\right)$$

(3.8)

The equation in the z-direction:

$$\left(\frac{\delta\sigma_{zz}}{\delta z}+\frac{\delta\tau_{xz}}{\delta x}+\frac{\delta\tau_{yz}}{\delta y}\right)+\rho g_z=\rho\left(\frac{\partial w}{\partial t}+u\frac{\partial w}{\partial x}+v\frac{\partial w}{\partial y}+w\frac{\partial w}{\partial z}\right)$$

(3.9)

And these three equations are the Navire – stocks equations.

3.2.2.3 Conservation of energy:

First law of thermodynamics: rate of change of energy of a fluid particle is equal to the rate of heat addition plus the rate of work done.

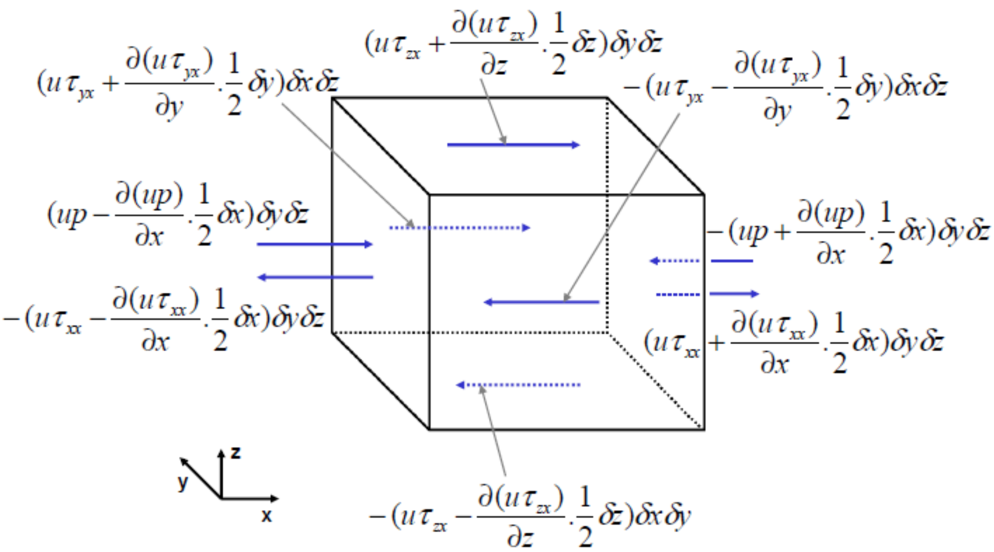


Figure 3.6 Energy fluxes associated with an infinitesimally small, moving fluid element. For simplicity, only the fluxes in the x direction are shown

The final form of the energy equation is:

$$\rho\frac{D}{Dt}\left(e+\frac{V^2}{2}\right)=\rho\dot{q}_x+\frac{\partial}{\partial x}\left(k\frac{\partial T}{\partial x}\right)+\frac{\partial}{\partial y}\left(k\frac{\partial T}{\partial y}\right)+\frac{\partial}{\partial z}\left(k\frac{\partial T}{\partial z}\right)+\frac{\partial\left(u\tau_{xx}\right)}{\partial x}+\frac{\partial\left(u\tau_{yx}\right)}{\partial y}+$$
$$\frac{\partial\left(u\tau_{zx}\right)}{\partial z}+\frac{\partial\left(v\tau_{yy}\right)}{\partial y}+\frac{\partial\left(v\tau_{xy}\right)}{\partial y}+\frac{\partial\left(v\tau_{zx}\right)}{\partial z}+\frac{\partial\left(w\tau_{zz}\right)}{\partial z}+\frac{\partial\left(w\tau_{yz}\right)}{\partial y}+\frac{\partial\left(w\tau_{xz}\right)}{\partial x}+\rho\vec{f}.\vec{V}$$

(3.10)

3.2.3 Types of algorithms used

ANSYS fluent provides four segregated types of algorithms: SIMPLE, SIMPLEC, PISO, and (for time-dependent flows using the Non-Iterative Time Advancement option (NITA)) Fractional Step (FSM). These schemes are referred to as the pressure-based segregated algorithm. Steady-state calculations will generally use SIMPLE or SIMPLEC, while PISO is recommended for transient calculations.

3.2.3.1 SIMPLE vs. SIMPLEC

In ANSYS fluent, both the standard SIMPLE algorithm and the SIMPLEC algorithm are available. SIMPLE is the default, but many problems will benefit from using SIMPLEC, particularly because of the increased under-relaxation that can be applied, as described below.[10]

For relatively uncomplicated problems (laminar flows with no additional models activated) in which convergence is limited by the pressure-velocity coupling, you can often obtain a converged solution more quickly using SIMPLEC. With SIMPLEC, the pressure-correction under-relaxation factor is generally set to 1.0, which aids in convergence speed-up. In some problems, however, increasing the pressure-correction under-relaxation to 1.0 can lead to instability due to high mesh skewness. For such cases, you will need to use one or more skewness correction schemes, use a slightly more conservative under-relaxation value (up to 0.7), or use the SIMPLE algorithm.

For complicated flows involving turbulence and/or additional physical models, SIMPLEC will improve convergence only if it is being limited by the pressure-velocity coupling. Often it will be one of the additional modeling parameters that limits convergence; in this case, SIMPLE and SIMPLEC will give similar convergence rates.[10]

The algorithm may be summarized as follows:

The basic steps in the solution update are as follows:

1. Set the boundary conditions.
2. Compute the gradients of velocity and pressure.
3. Solve the discretized momentum equation to compute the intermediate velocity field.
4. Compute the uncorrected mass fluxes at faces.
5. Solve the pressure correction equation to produce cell values of the pressure correction.

3.3. Flow classifications:

The flow of any fluid has some classifications such as; velocity pattern, Change of Density, Statistically Steady Flow or Unsteady and Inviscid and Viscous Flow

3.3.1. Velocity pattern: Turbulent, laminar flow.

To understand and compare between turbulent and laminar let us assume a free-stream flow along the x-axis with uniform velocity U. If we follow the traces made by several particles in the fluid we would expect to see parallel lines these lines are parallel and follow in the direction of the average velocity, and the motion of the fluid seems to be “well organized”, then this flow is called laminar [11].

Then the velocity vector will be:

$\vec{q} = (u, v, w)$ (4.12)

Then for this steady state flow the velocity vector will be:

$\vec{q} = (U, 0,0)$ (4.13)

Where U is the velocity into the x direction, and q for the velocity vector.

Assuming in the second case have the same average speed (U_{av})in the flow, but in addition to this average speed the fluid particles will momentarily move into the other directions. The fluid is then called turbulent. Assuming the average velocity for laminar and turbulence flow is the same. Also, in this two-dimensional case the flow is time dependent everywhere and average velocities into y and z directions are zero, the velocity vector then becomes:

$\vec{q} = (U_{av} + u, v, w)$. (4.14)

Where $V_{av} = W_{av} = 0$ (4.15)

Where u, v, w are the perturbation into the x, y, and z directions.

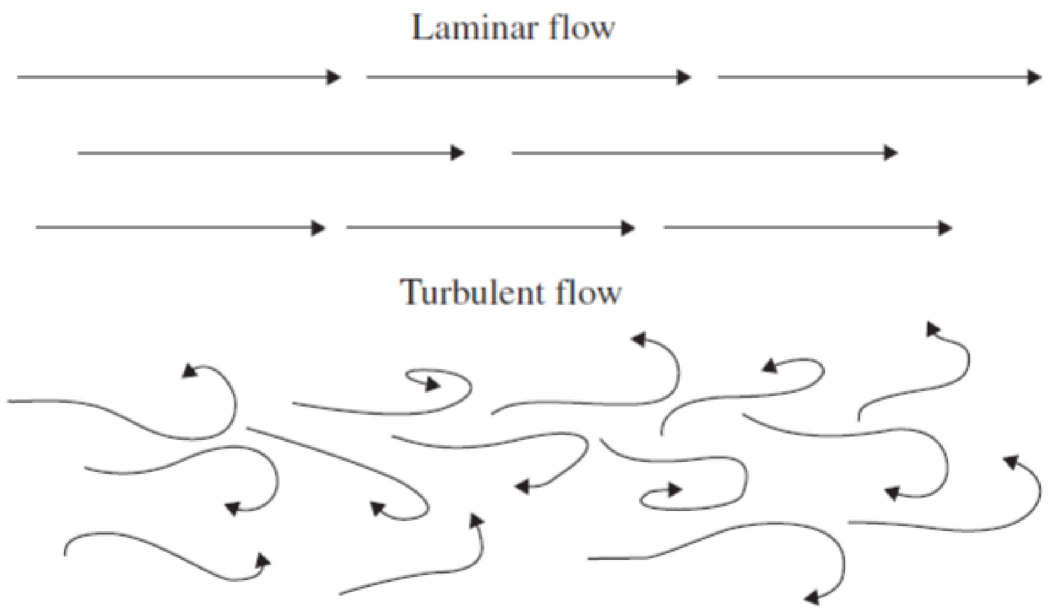


Figure 3.7 Schematic descriptions of laminar and turbulent flows with the same average velocity.

3.4 Visualization of fluid flow:

The quantitative and qualitative information of fluid flow can be obtained through sketches, photographs, graphical representation and mathematical analysis. However, the visual representation of flow fields is very important in modeling the flow phenomena. There is three sets of curves describes the fluid motion such as; Pathline, streakline, and streamline[11].

3.4.1 Pathlines:

Pathline or a particle path is a curve describing the trajectory of a fluid element .Pathlines are obtained in the Lagrangian approach by an integration of the equations of dynamics for each fluid particle. And can be produced in the laboratory by marking the fluid particle and taking time exposure photograph of its motion[11].

3.4.2. Streakline:

In many cases of experimental flow visualization, particles are introduced into the flow at a fixed point in space. The line connecting all of these particles is called a streak line. Here, the attention is focused to a fixed point in space (i.e. Eulerian approach) and identifying all fluid particles passing through that point. These lines are laboratory tool rather than analytical tool. They are obtained by taking instantaneous photographs of selected particles that have passed through a given location in the flow field [11].

3.4. 3 Streamline:

These are the lines drawn in the flow field so that at a given instant, they are tangent to the direction of flow at every point in the flow field. Mathematically, these lines are obtained analytically by integrating the equations defining lines tangent to the velocity field. In a two-dimensional flow field as shown in the figure 3.8, the slope of the streamline is equal to the tangent of the angle that velocity vector makes with x-axis as shown in the figure below:

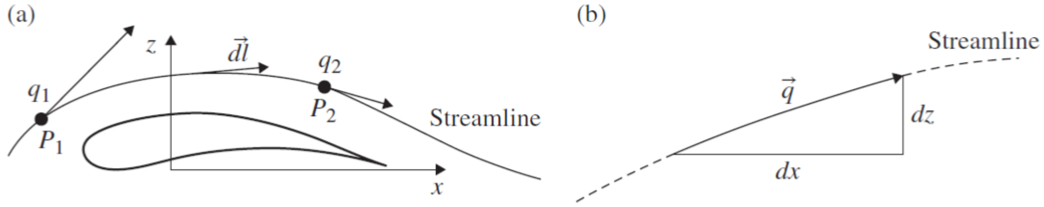


Figure 3.8

Where q the tangent velocity vector, l the streamline

When q is tangent to the streamline dl therefore:

$$\vec{q}.\vec{dl} = 0 \qquad (4.16)$$

4

Chapter Four

Modeling and simulation results

4.1 Project Design

4.2 Model Creaction

4.3 Processes

4.4 Results

4.5 Conclusion

4.6 Recommendations

4.1 Project Design

A drown model on solid works was taking from GrabCAD site, for Skoda Octavia 2012 , the model imported to solid works in order to make adjustments including cutting the front and back parts, to have only the cabin of the vehicle to make it easier for calculation on ANSYS software to fit the requirements of the designed system ,four cases were created on solid works software ,each case has a different dimension design ,the holes crated randomly , after making adjustments on solid works , the next step was importing the model on ANSYS, software, after done form calculation on ANSYS software, it gives the right and the appropriate design purpose to make the heat exchange due to high ambient temperature between inside and under the vehicle.

, .

Figure 4. 3 Random Holes on Surface of the Vehicle

4.2 Model creation

Model1

In model 1, the design of the system in the cabin of the vehicle consist of two holes in the celling ,one in the middle of the ceiling and the other in the back of the celling ,another two holes in floor one in the front of the vehicle and the other in the back, the diameter of each hole is 5cm, the dimension of the design shown in the fig below:

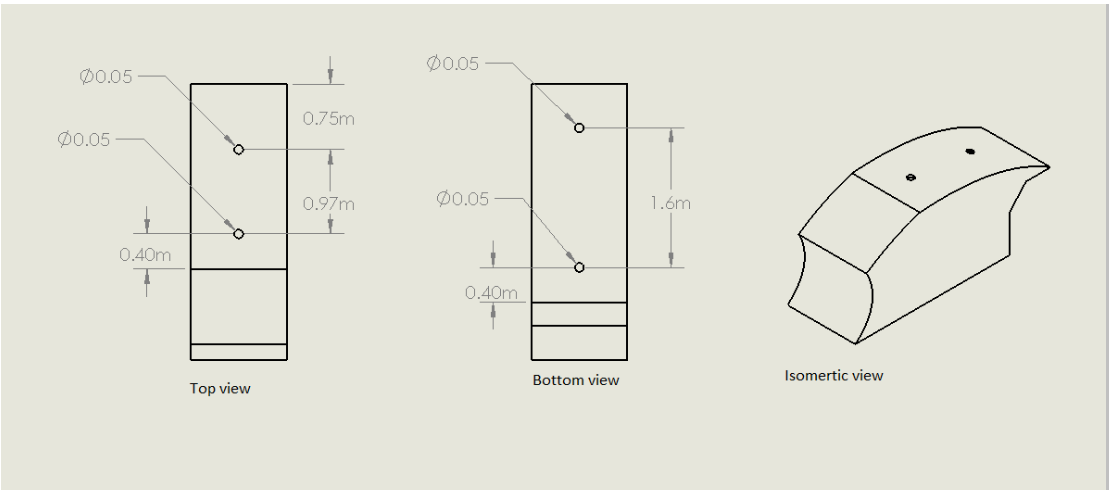


Figure 4. 2 Dimensions for model 1

Model 2

In model 2 , the design of the system in the cabin of the vehicle consist of two holes in the middle of the celling parallel to each other, and another two holes in floor in the back of the vehicle also parallel to each other ,the diameter of each hole is 5cm, the dimension of the design shown in the fig below: .

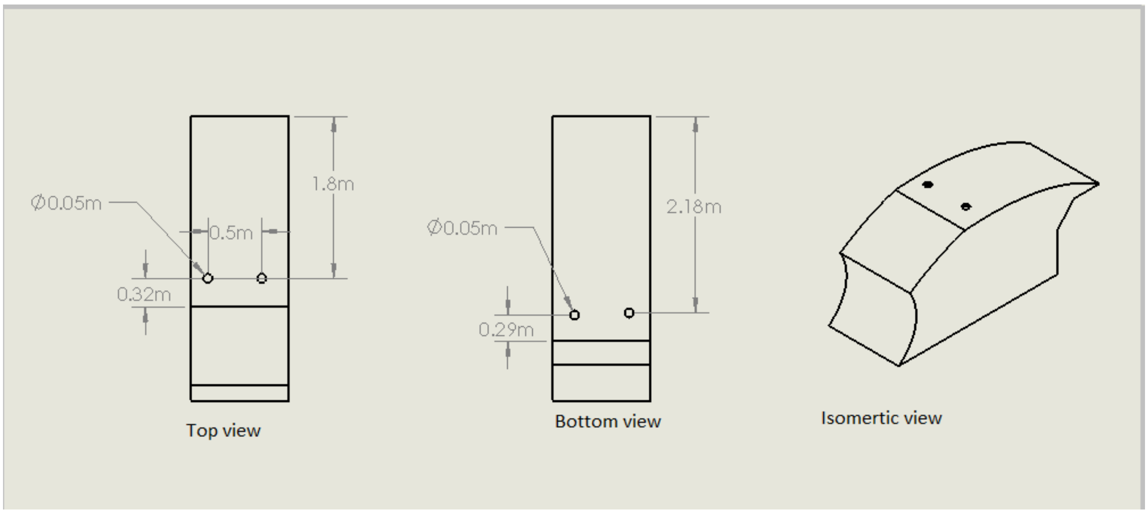


Figure 4. 3 Dimensions for model 2

Model 3

In model 3, the design of the system in the cabin of the vehicle very similar to case2,it consist of two holes in the ceiling parallel to each other, and another two holes in floor in the back of the vehicle also parallel to each other, the diameter of each hole is 7cm , the dimension of the design shown in the fig below:

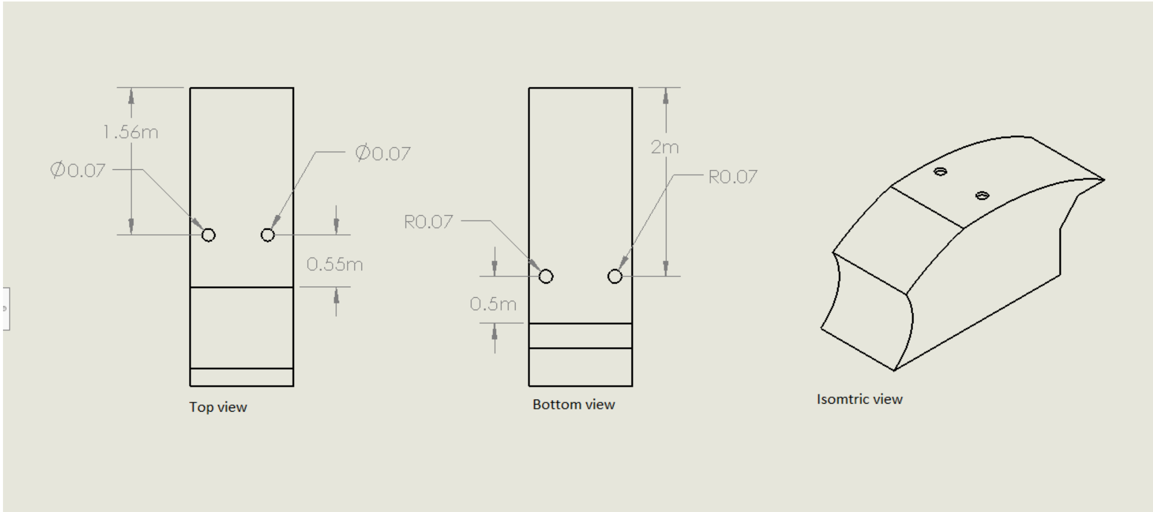


Figure 4. 3 Dimensions for model 3

Model 4

In model 4, the design of the system in the cabin of the vehicle was quite different than the other models above , a drown shape in the in the middle of the ceiling and the other one in the back of the floor, the dimension of the design shown in the fig below.

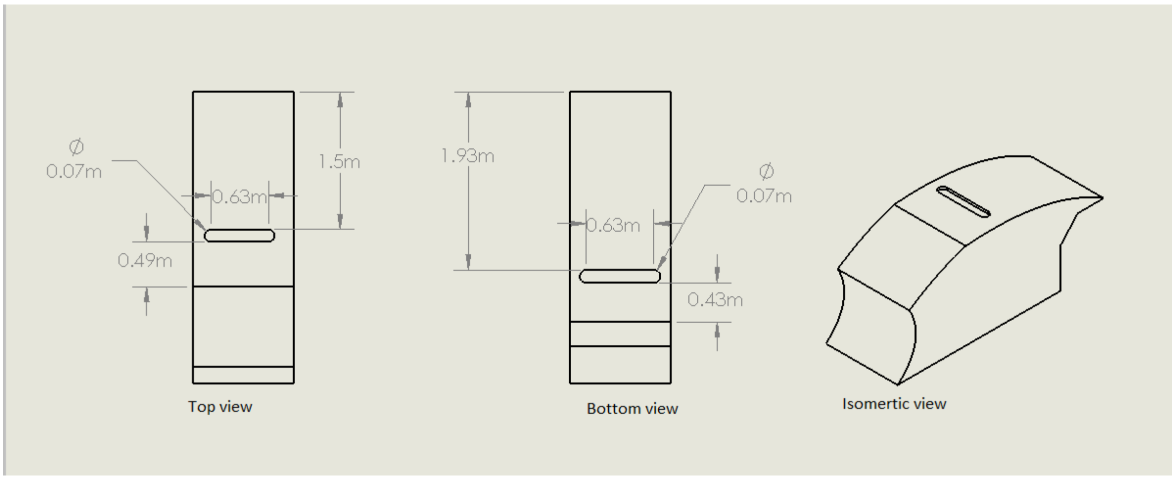


Figure 4. 5 Dimensions for model 4

4.3 Processes

In ANSYS software 4 steps must be done in older to get the results which is meshing part, boundary condition, setup and finally the result part, each step will be explained.

- Meshing

One of the most important parts on which the solution of the flow equations depends on, is the selection of an appropriate mesh. The mesh represents how a computational domain it is discretized into a finite number of control volumes. Obviously, the accuracy of the solution improves with the increase in the number of control volumes, a fine enough mesh is required to ensure that the solution is accurate, for this kind of studies, using a small enough mesh size will give good results. The model moved to the mesh part, and a subtle mesh was created, a fine mesh was chosen to a relevance center as shown in Fig 4.6, also a high smooth was chosen to the model as shown in Fig4.7, the number of max size was made small 8e-003 as shown in Figure 4.8 After that, mesh well be done and ready as shown in Figure 4.9.

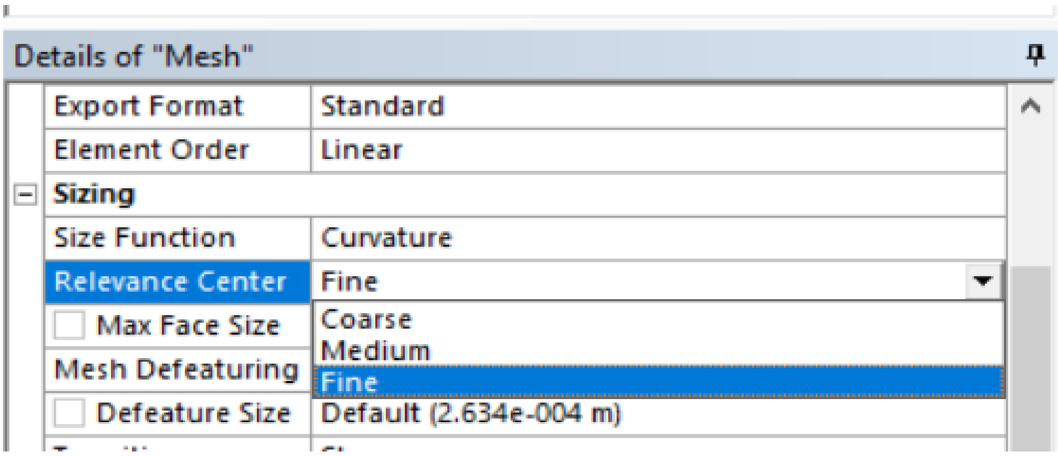


Figure 4.6 Fine mesh

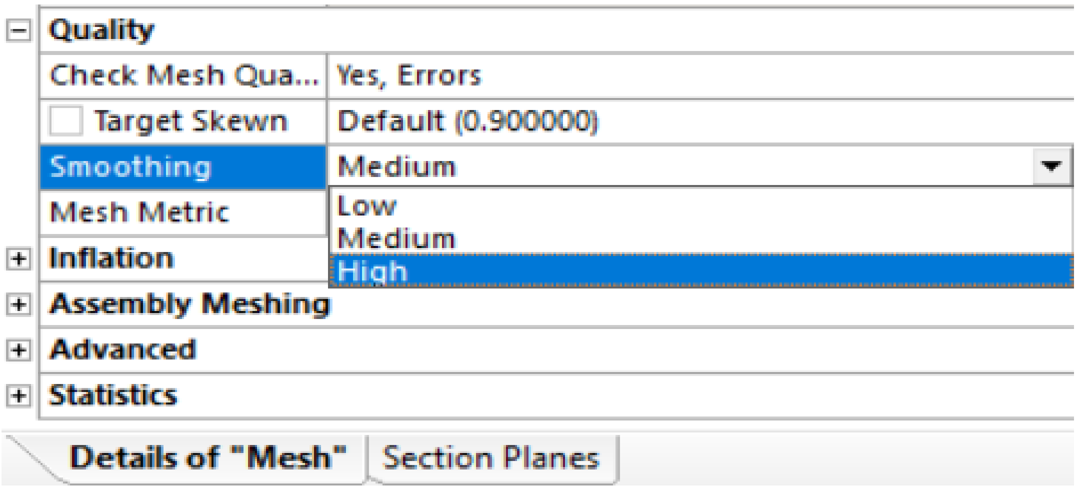


Figure 4.7 Smoothing

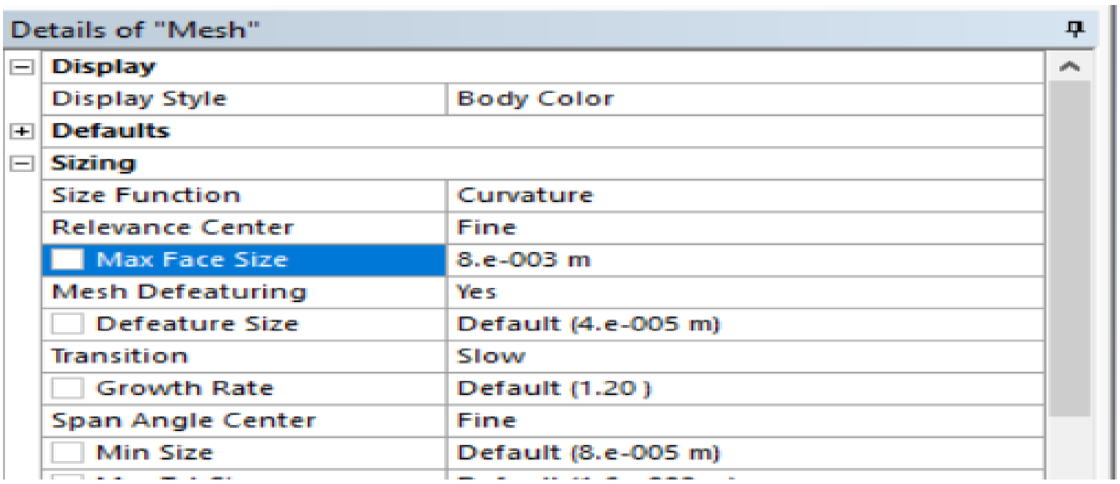


Figure 4.8 Max Face Size

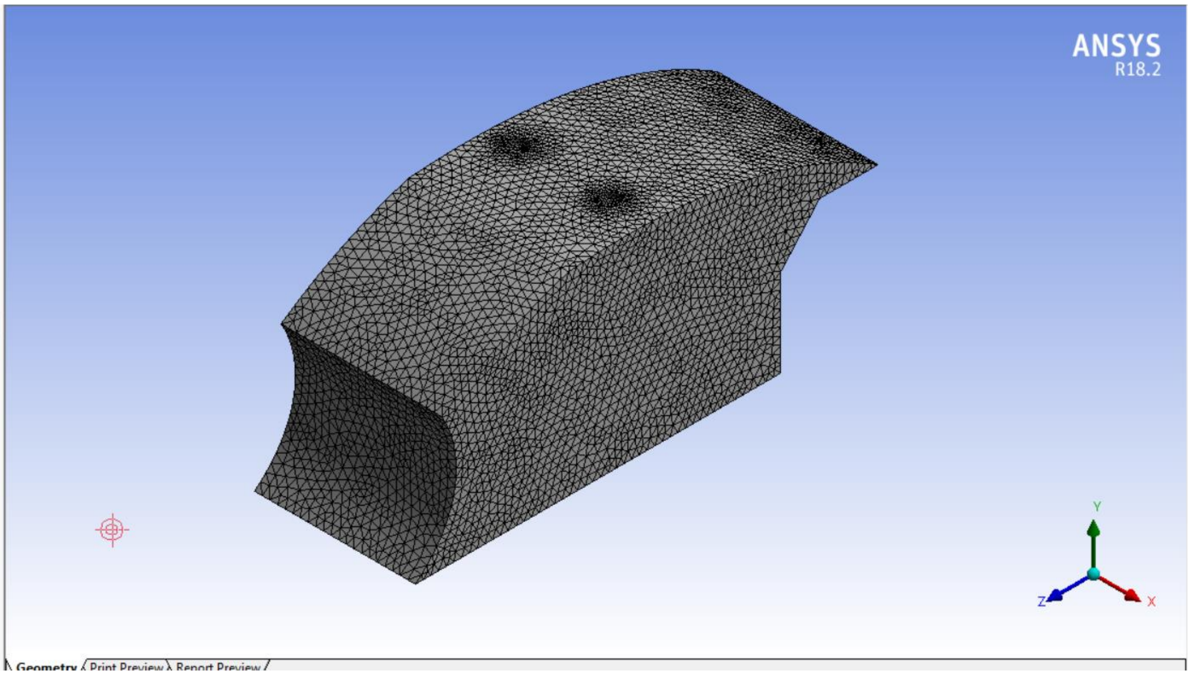


Figure 4.9 The Cabin After Meshing

- Boundary condition

It is crucial in CFD simulations that proper and accurate boundary conditions are specified. Boundary conditions indicate how the user interprets the specific physical phenomena into a CFD code. Thus, detailed information about boundary conditions is important in properly prediction of CFD results. If physical phenomenon is not properly incorporated through the boundary conditions, then the corresponding analysis can give false result and its interpretation can be inaccurate. The named boundary condition shown in figure 4.10, were chosen to specify the boundary condition of the vehicle, the boundary were named as follow:

- The two holes above the vehicle were named outlet1, outlet2.
- The two holes under the vehicle were named inlet1, inlett2.
- The wall of the vehicle were named domain.

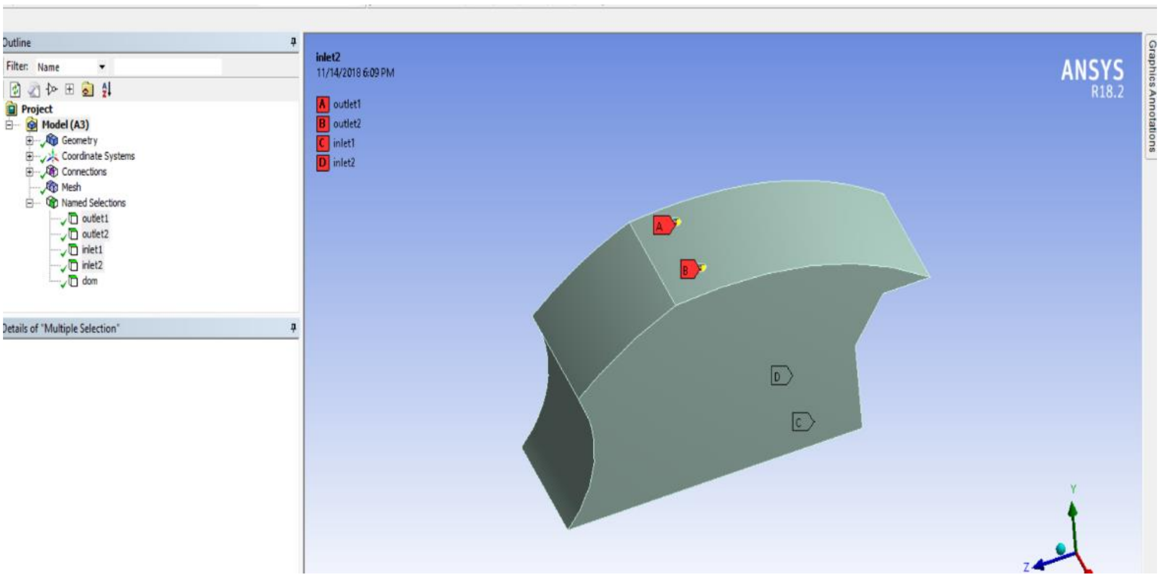


Figure 4.10 The named boundary conation

- Setup

Fluid dynamics setup is an important part of CFD analysis because deal with the actual flow equations that are solved numerically. In addition to the correct settings in the software , and using the right numerical procedure gives results that are more accurate. In setup firstly, $k-\varepsilon$ was chosen, as shown in figure 4.11, due to its good convergence rate and relatively low memory requirements, the energy equation was putting on for the boundary condition ,the temperature of inlet1 and inlet2 was set 20 C°, for outlet 2 and outlet1, the temperature set 39 C° and the temperature of domain set as 60 C°, for the fluid martial the air was set as a fluid, and for solid martial the aluminum was chosen as shown in Figure 4.12 .

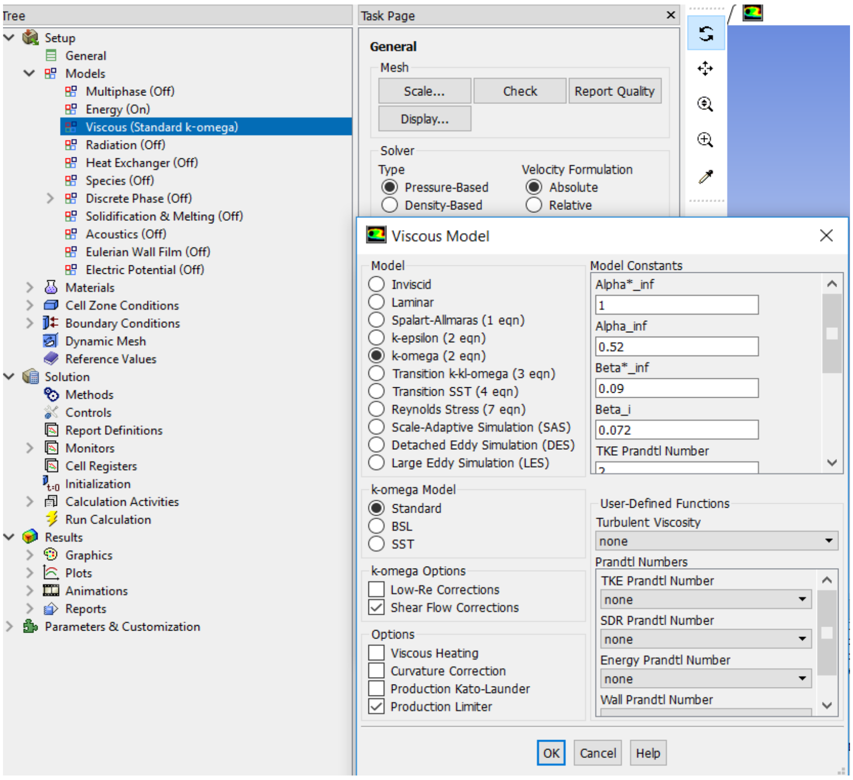


Figure 4.11 the model was chosen

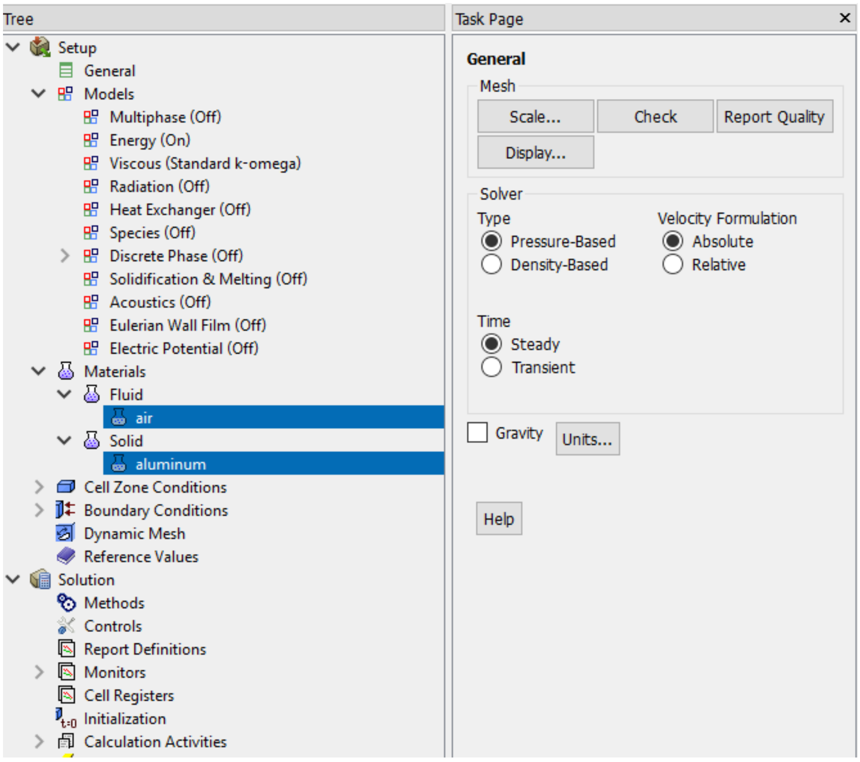


Figure 4.12 Aluminum was chosen

After all these steps, the model will be ready to initialization and calculation, the result will appear on the result section.

4.4 Results

In result section ,the four cases will be discuss and compare to each other, the result will be focusing on the temperature, three planes were created inside the model in order to see the loss of heat inside the cabin, the location of the planes were created in different location, also a streamlines plane was created to see the behave of the streamlines in the cabin ,and finally a velocity vector plane was created,

Case1

In case1 , the design of the system for the cabin was explained in the model1, the plane in the results show that, the temperature inside the cabin is around 41 C° as Shown in Figure 4.13, that means, the temperature decrees from 60 C° to around 41 C°, the Figure 4.14 below shows the behave of streamlines, it shows how the streamlines came from under the vehicle and then spread inside the cabin, Figure 4.15 shows the velocity vector.

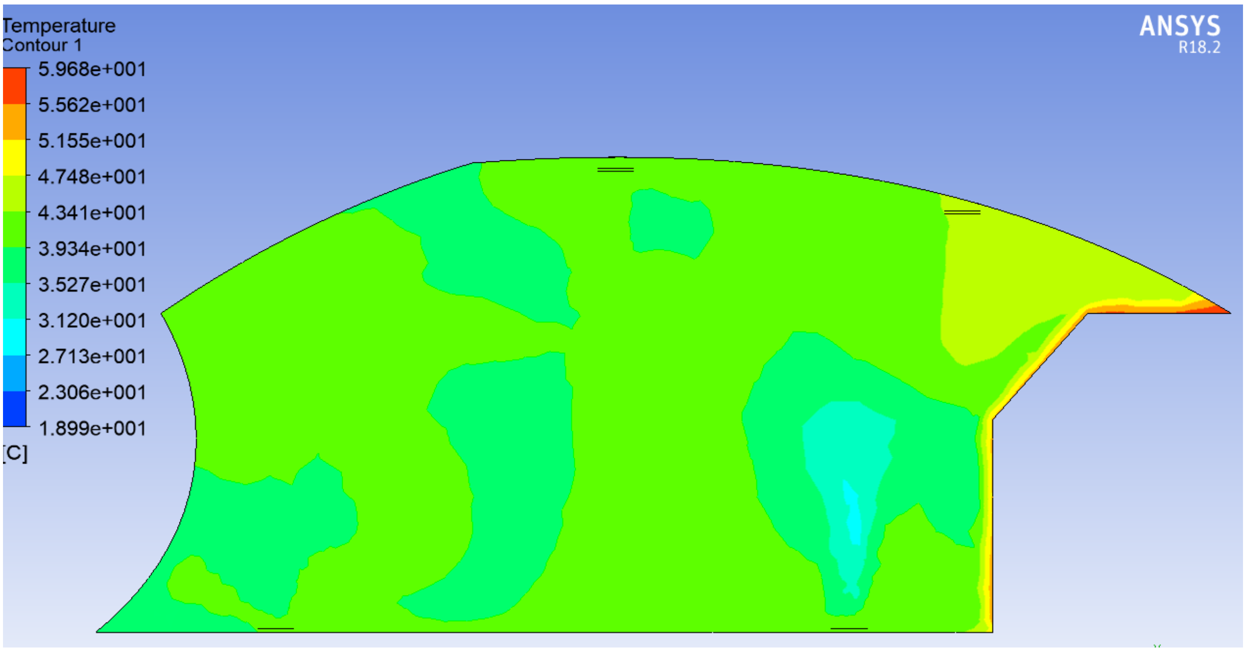


Figure 4.13 the temperature inside case1

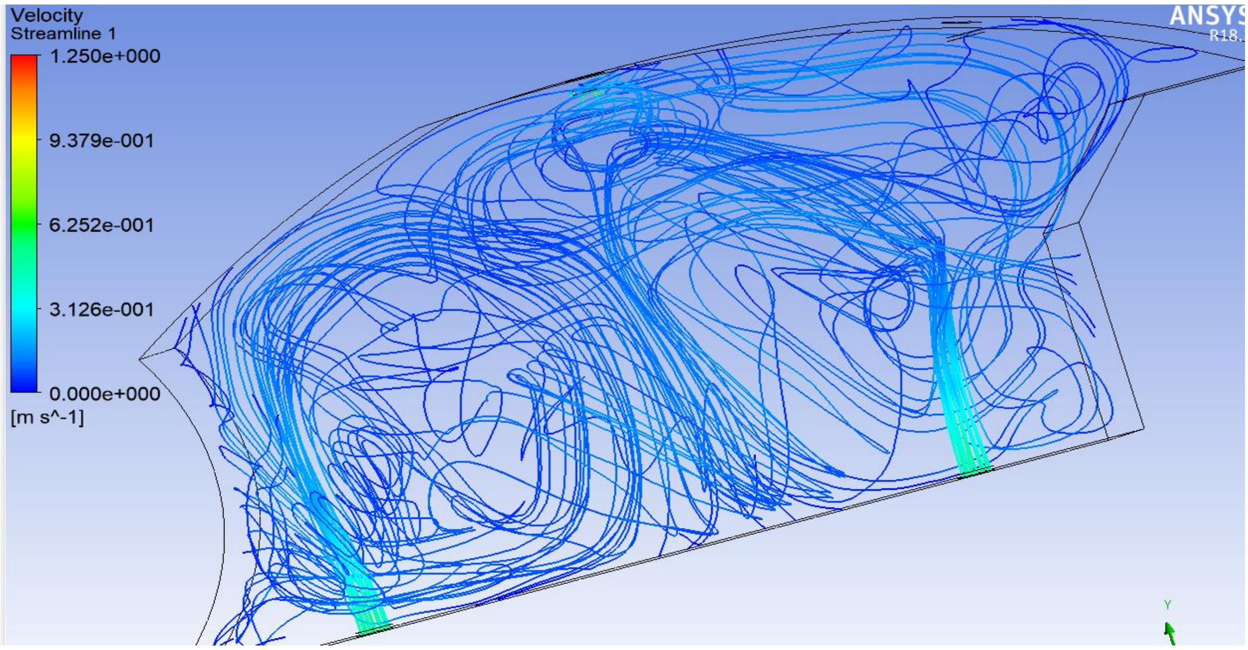


Figure 4.14 The streamline inside case1

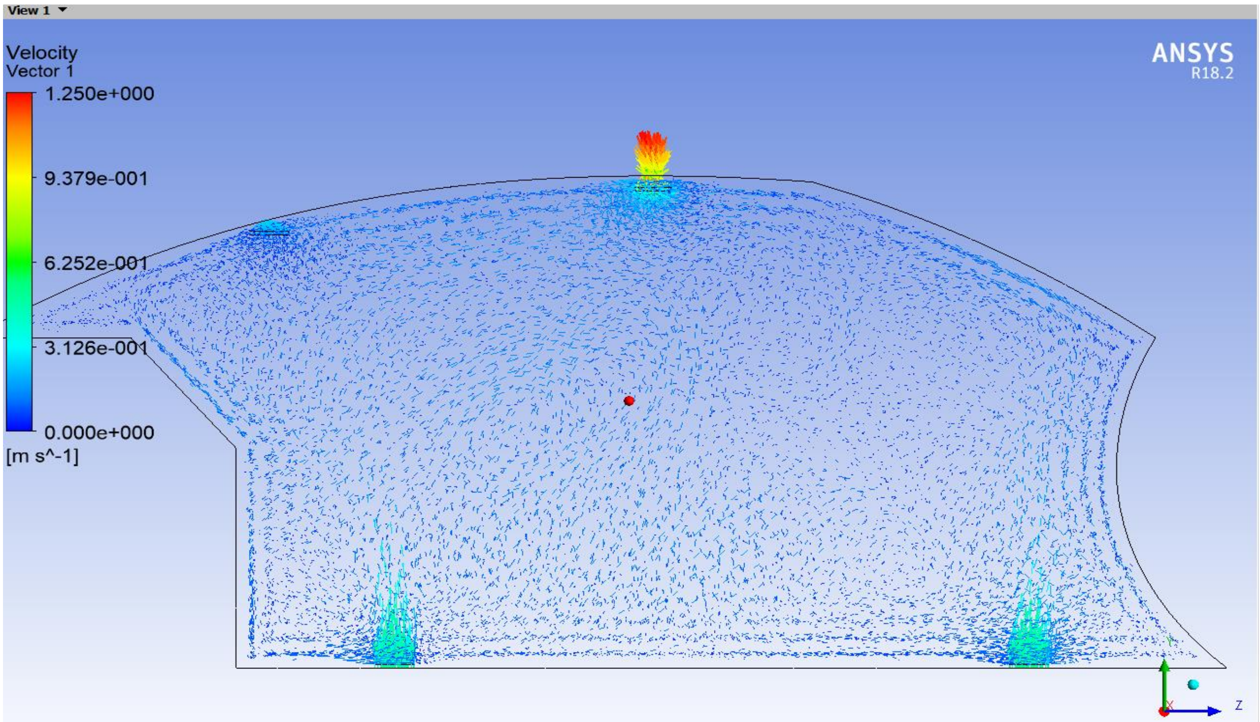


Figure 4.15 the velocity vector inside case1

Case2

In case 2, the design of the system for the cabin was also explained in the model 2, the results show that the temperature inside the cabin is around 46 C° that means the temperature decrees from 60 C° to around 46 C° as shown Figure 4.16 ,the Figure 4.17 below shows the behave of streamlines it shows how the streamlines came from under the vehicle and then spread inside the cabin, Figure 4.18 shows the velocity vector.

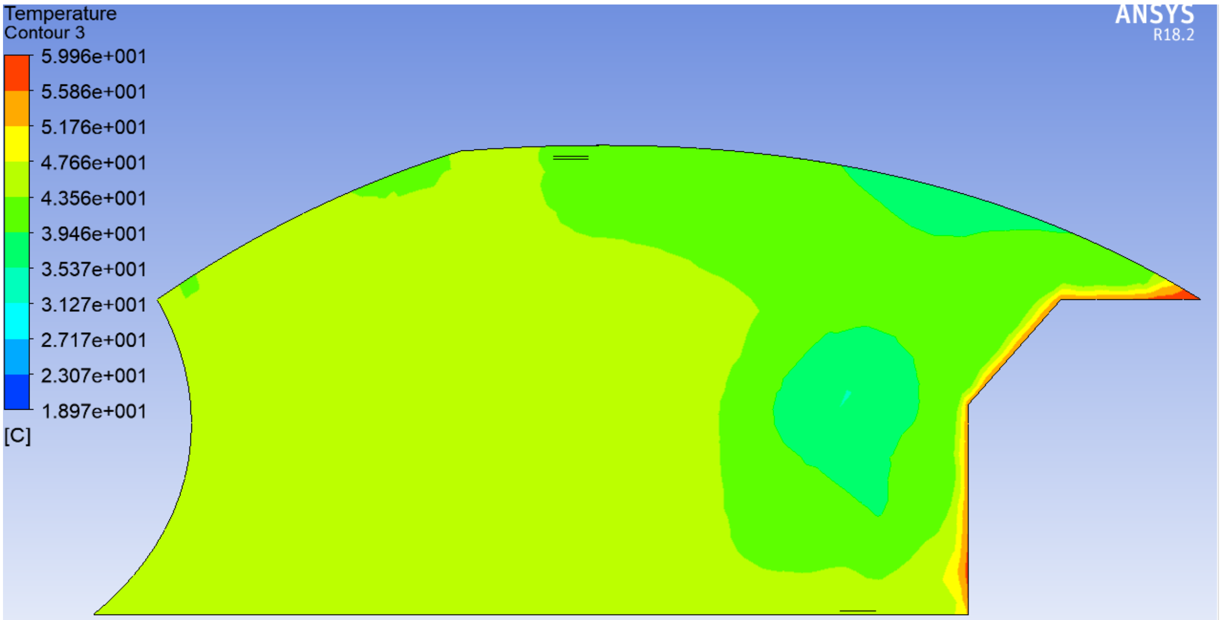


Figure 4.16 The temperature inside case2

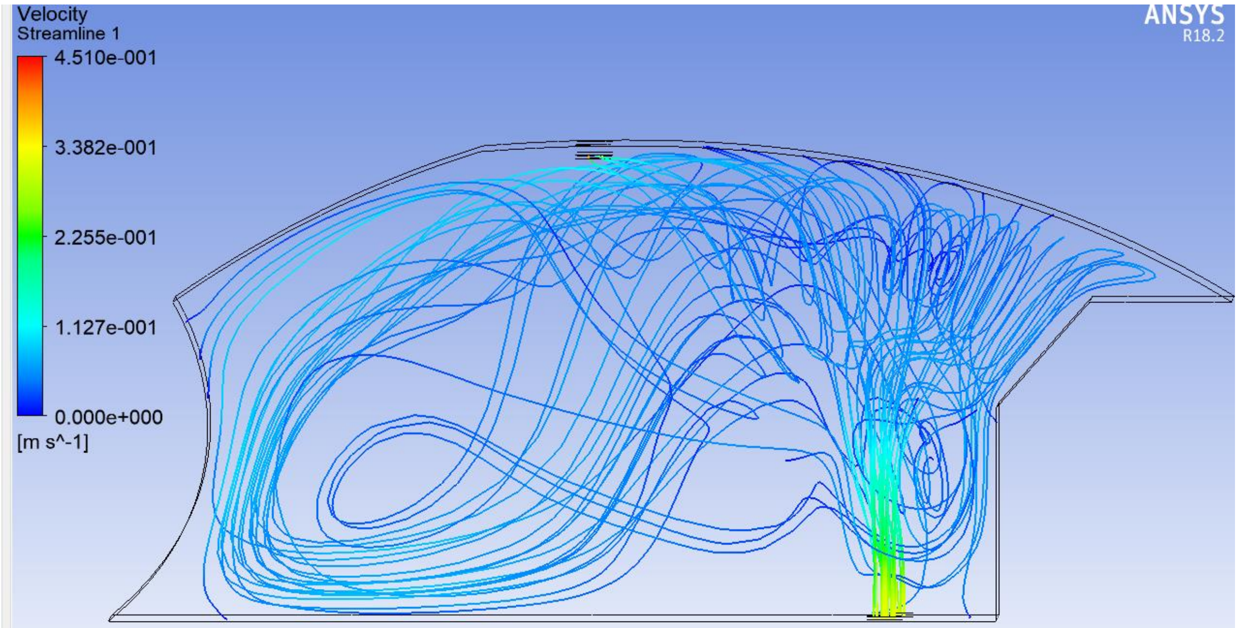


Figure 4.17 The streamline inside case2

1

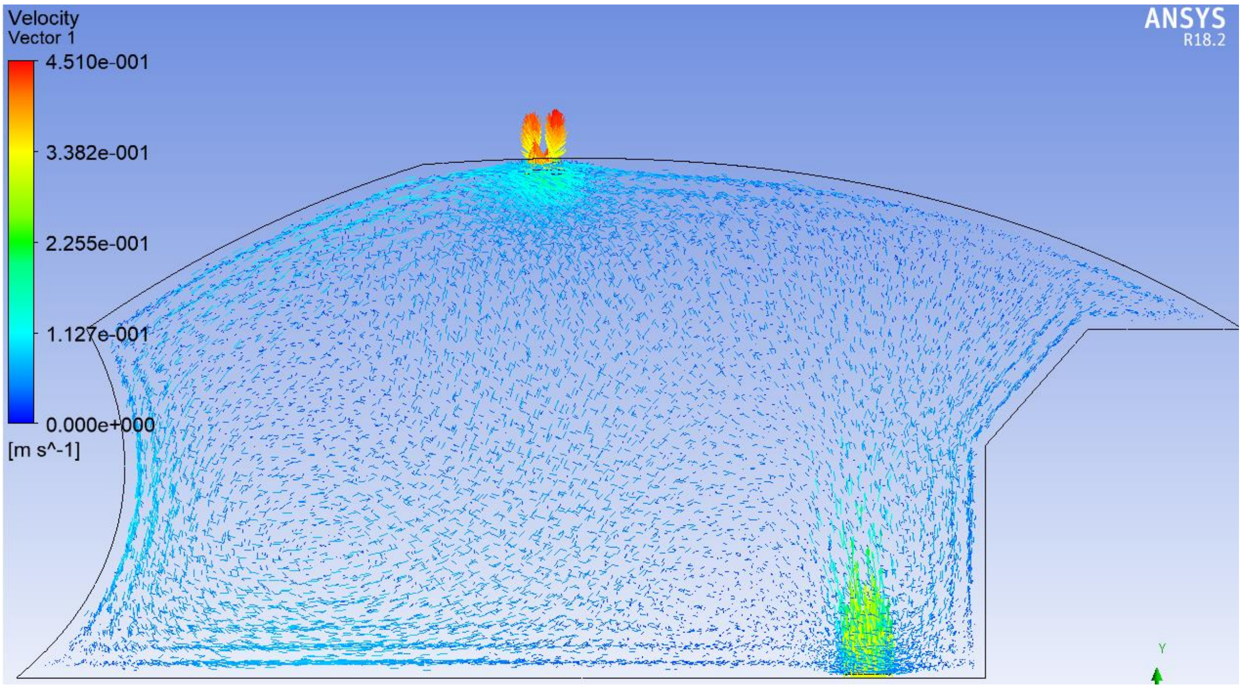


Figure 4.18 The velocity vector case2

Case 3

In case 3, the design of the system in the cabin is very similar to case 2, the design explained in the model 3, the results show that the temperature inside the cabin is around 38 C° that means the temperature decrees from 60 C° to around 39 C° as shown Figure 4.19 ,the Figure 4.20 below shows the behave of streamlines, it shows how the streamlines came from under the vehicle and then spread inside the cabin, Figure 4.21 shows the velocity vector.

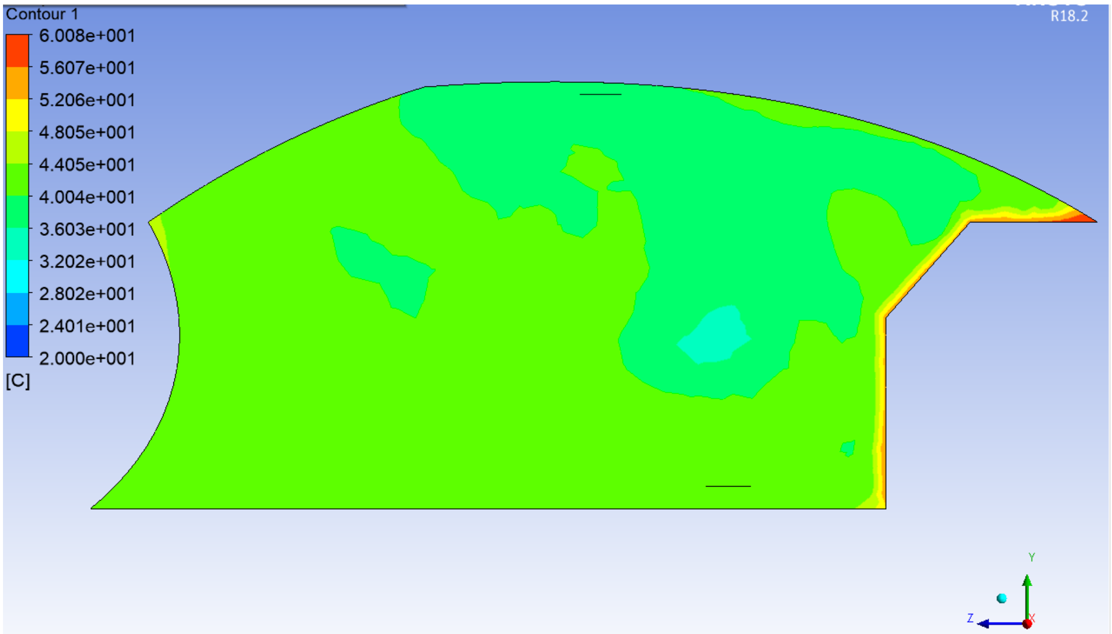


Figure 4.19 The temperature inside case3

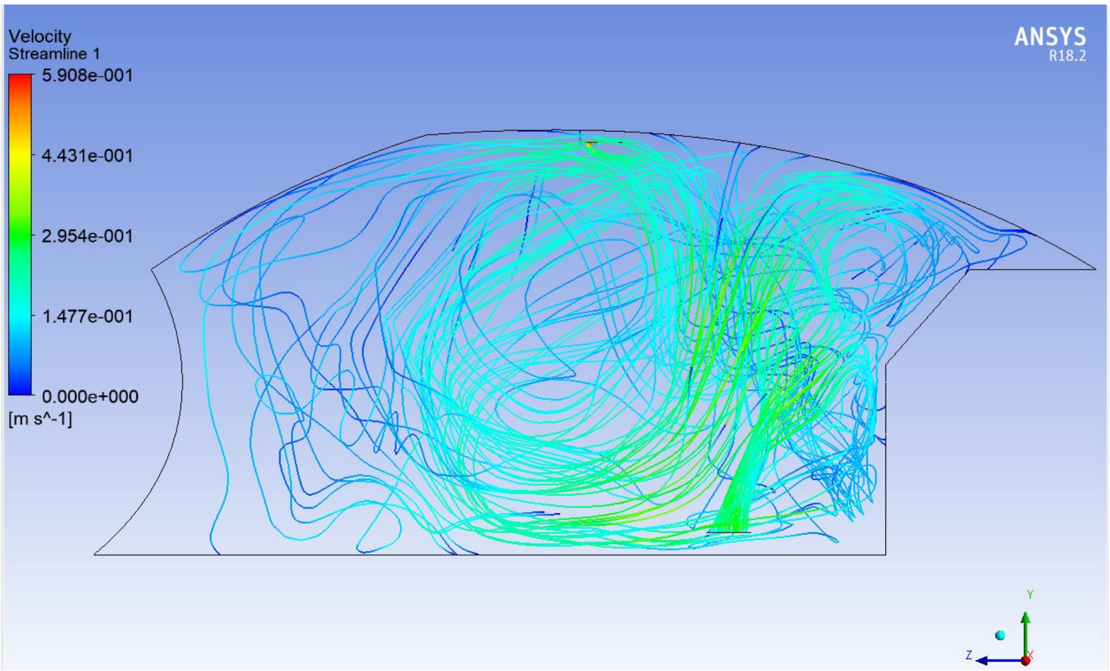


Figure 4.20 The streamline inside case3

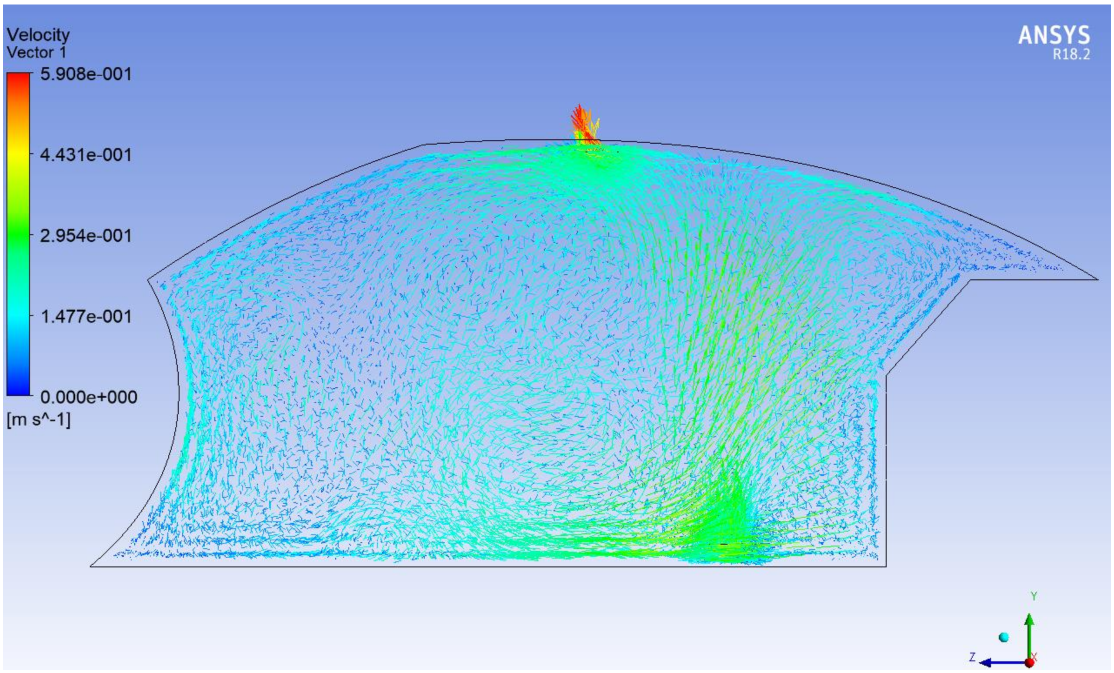


Figure 4.21 The velocity vector case3

Case 4

In case 4 , the design of the system was quite different than the other models, the design explained in model 4, the results show that the temperature inside the cabin is around 33 C° that means the temperature decrees from 60 C° to around 33 C° as shown Figure 4.22 ,the Figure 4.23 below shows the behave of streamlines it shows how the streamlines came from under the vehicle and then spread inside the cabin, Figure 4.24 shows the velocity vector.

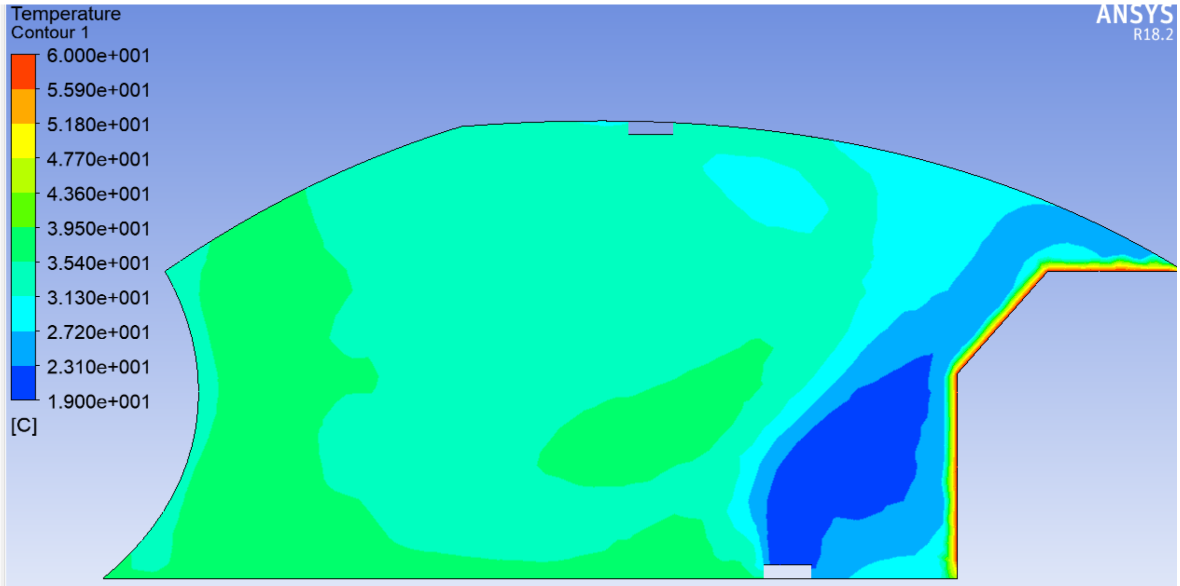


Figure 4.22 The temperature inside case41

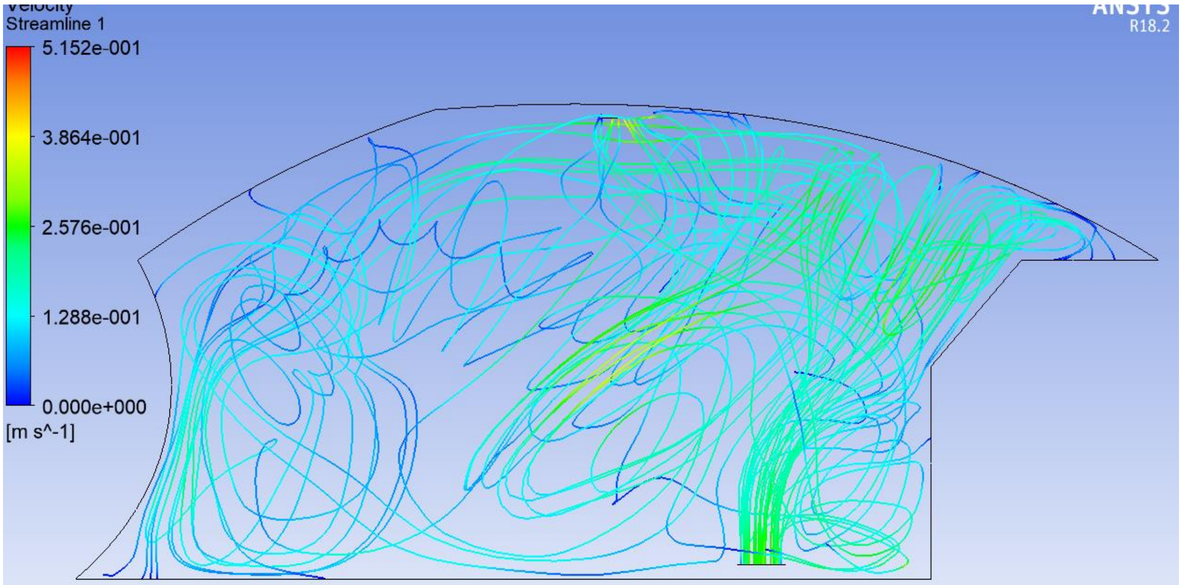


Figure 4.23 The streamline inside case4

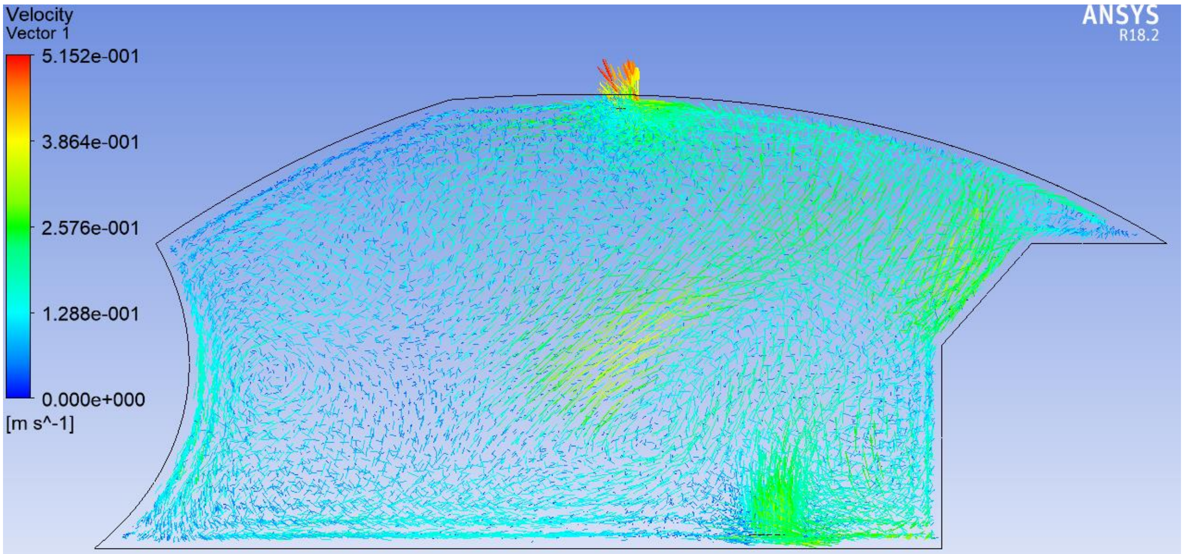


Figure 4.24 The velocity vector case4

4.5 Conclusion

The high increasing of temperature inside the cabin of the vehicle, lead us to suggested solution , the solution was to design a safety system works on passive natural ventilation in vehicles when its parked , the principle of the system depends on convection method, a simulation for a natural convection inside a cabin of a vehicle was perfumed , to reduce the temperature inside the cabin, the four cases show that the temperature decreased inside the cabin, case 4 shows the best increasing of temperature was due to the a heat exchange between the temperature of the shadow which is around 20 C° and the temperature of the cabin which is around 60 C°, the loss of the heat was from 20 C° to 27 C° and that was depending on the design of the system on the cabin.

4.6 Recommendation

After completing our simulation design project , the main recommendation of this project are as follow:

- To create a practical real model to reflect our results reached by this project .
- Then to measure the temperature inside this model and to compare it with the simulation rustles.
- To make the design more suitable and safe by using nets for the holes .
- One recommendation for the faculty of engineering at PPU , is to add the ANSYS software as required subject for the mechanical engineering .

References:

[1] *C. Goh, L. Kamarudin, S. Shukri, N. Abdullah, and A. Zakaria, "Monitoring of carbon dioxide (CO 2) accumulation in vehicle cabin," in Electronic Design (ICED), 2016 3rd International Conference on, 2016, pp. 427-432.*

[2] *E. A. Q. Dr.Mohammad.G.Q , Eng.Nayef.T, "smart automobile air conditioning and ventilation system " May 2017.*

[3] *D. Yingmeng, J. Wang, W. Huang, and Z. Fan, "Simulation and analysis on heat transfer and pre-cooling characteristics of new solar power vehicle parking ventilation system ",in Transportation Electrification Conference (ITEC), 2015 IEEE International, 2015, pp. 1-4.*

[4] *Z. Li, H. Xu, G. Tan, Z. Yu, Z. Yang, Z. Li, et al., "Pushing-pulling based vehicle parking ventilation cooling characteristics analysis," in Transportation Electrification Conference (ITEC), 2015 IEEE International, 2015, pp. 1-6.*

[5] *E. amena.a.t. (2010). Natural ventilation in buildings. Available: https://mirathlibya.blogspot.com/2010/09/blog-post_22.html?m=1*

[6] *J. H. Lienhard, A heat transfer textbook :Courier Corporation, 2013.*

[7] *http://www.figes.com.tr/english/ANSYS/ANSYS.php.*

[8] *John Wiley & Sons , Joseph Katz, Automotive aerodynamics San Diego State University,USA,2016*

[9] *Y. A. Cengel, Introduction to thermodynamics and heat transfer: McGraw-Hill New York, 1997.*

[10] *ANSYS.INC, "Choosing the Pressure-Velocity Coupling Method," pp. http://www.afs.enea.it/project/neptunius/docs/fluent/html/ug/node785.htm?fbclid=IwAR3PaHmOzyR4hnnv18WUrJ8gc_Mz-5y5yGdmUU4rtDg2GwxLJQ1dW-DE3TM, 2009.*

[11] *Katz and A. Plotkin, Low-speed aerodynamics vol. 13: Cambridge university press, 2001.*

End of the project