International Journal for Ignited Minds (IJIMIINDS)

CFD Analysis for the Cooling Vent in Automobile

Vijay Kumar M[a], Dr. Basawaraj[b] & Dr. B R Shridhar[c]

[a]Asst Prof, Department of Mechanical, S.E.A.C.E.T, Bengaluru, India.
[a]Associate Prof, Department of A P T, VTU, CPGS, VIAT, Chikkaballapur, India.
[c]Prof, Department of Mechanical, S.E.A.C.E.T, Bengaluru, India.

ABSTRACT

Now a days the aerodynamics is the last frontier in which the race teams try to gain those precious seconds and levels of grip. Teams one coming up with new innovations which then will give them the edge. For an automotive engineer it is vital to know the basics in aerodynamics, because on in the commercial side the cars are made externally aerodynamic in order to achieve the lowest possible fuel consumption, and internally for more comfortable design. As the rules are very strict regarding the engine, power train, etc. what is left is the external and internal aerodynamics. Only few teams have incorporated external aerodynamics in their design and some of those teams have been quite successful. The main reason why the teams haven't spent time in developing the internal aerodynamics in the car cabin is because of lack of focus in getting more efficient cooling system. In this paper, basics of aerodynamics will be presented briefly, along with the basics of an internal cooling system and cooling vent design. Different designs of cooling vent are evaluated, from the point of view of their suitability to meet the desired requirements.

Keywords *– Airflow, HVAC, Aerodynamics, Power Train, Cooling System.*

1. INTRODUCTION

Human comfort in cars is of prime importance nowadays, in which thermal comfort plays an important role. With the rapid development of technology and increasing demands by customers, the climate control of the passenger cabin has to be taken into account in any vehicle development process. To enhance the competitive ability of an automobile to the satisfaction of customer's requirement automotive thermal comfort is of crucial importance. However, the comfort level being subjective it is hard to set definitive levels. The only controlling measures are airflow velocity, cabin temperature and relative humidity. Hence continuous research and investigation is being done to achieve more and more thermal comfort to the passengers.

Improving air conditioning performance and occupant thermal comfort requires an understanding of the fluid motion prevailing in the cabin for required ventilation setting and passenger loading. HVAC (heating ventilation and air conditioning) system being the heart of air conditioning should be properly designed and packaged for a particular car. The size of the air-conditioning system is related to the peak thermal load in the vehicle.

1.1 Modes of Heat Transfer

Thermal analysis of a passenger compartment involves not only geometric complexities but also strong interactions between airflow and the three modes of heat transfer, namely heat Conduction, Convection and thermal radiation. Conduction is a mode of transfer of heat which occurs when a temperature gradient exists in a body or when two or more bodies are in the same phase.

The heat transfer rate per unit area is proportional to the normal temperature gradient:

$$\frac{\delta T}{\delta x} \sim \frac{q}{A}$$

When the proportionality constant is inserted:

$$q = -kA\frac{\delta T}{\delta x}$$

Where q= Heat transfer rate

$\frac{\delta T}{\delta x}$ =Temperature gradient in the direction of flow

k=Thermal conductivity of the material.

A= Area under consideration.

The minus sign is inserted so that the second principle of thermodynamics will be satisfied. The equation is called the Fourier's law of Heat Conduction. Convection is the mode of heat transfer between any two different phases such as a solid and a liquid or a solid with a gas or vice versa. The overall effect of convection is expressed by Newton's law of Cooling.

$$Q = h*A*\Delta T$$

Where, q= Heat transfer rate
h=Convection Heat transfer Coefficient
ΔT=Temperature gradient between the two bodies
A=Area under consideration

Energy transfer can also take place without a medium this mode of heat transfer is called Radiation .The mechanism in this case is electromagnetic radiation. Thermodynamic considerations show that an ideal thermal radiator or a black body will emit energy at a rate proportional to the fourth power of the absolute temperature of the body.

$$Q_{emitted} = \sigma * T^4$$

Where **σ**=Stefan-Boltzmann Constant (5.669 x 10-8 W/m2.K4)

2. COMPUTATIONAL FLUID DYNAMICS (CFD)

Computational Fluid Dynamics (CFD) is a computer-based tool for simulating the behavior of systems involving fluid flow, heat transfer, and other related physical processes. It works by solving the equations of fluid flow (in a special form) over a region of interest, with specified (known) conditions on the boundary of that region.

2.1 Ansys ICEM CFD

Meeting the requirement for integrated mesh generation and post processing tools for today's sophisticated analysis, ANSYS ICEM CFD provides advanced geometry acquisition, mesh generation, mesh optimization, and post-processing tools. Maintaining a close relationship with the geometry during mesh generation and post-processing, ANSYS ICEM CFD is used especially in engineering applications such as computational fluid dynamics and structural analysis.

2.2 Meshing Modules

Hexa Mesh:

This ANSYS ICEM CFD semi-automated meshing module presents rapid generation of multi-block structured or unstructured hexahedral volume meshes. ANSYS ICEM CFD Hexa represents a new approach to grid generation where the operations most often performed by experts are automated and made available at the touch of a button. Blocks can be built and interactively adjusted to the underlying CAD geometry. This blocking can be used as a template for other similar geometries for full parametric capabilities. Complex topologies, such as internal or external O-grids can be generated automatically.

Hybrid Meshes:

Hybrid meshes can be created by several means: Tetra and Hexa meshes can be united (merged) at a common interface in which a layer of pyramids are automatically created at a common interface to make the two mesh types conformal. Good for models where in one part it is desired to have a "structured" hexa mesh and in another more complex part it is easier to create an "unstructured" tetra mesh. Hexa-Core meshes can be generated where the majority of the volume is filled with a Cartesian array of hexahedral elements essentially replacing the tetras. This is connected to the remainder of a prism/tetra hybrid by automatic creation of pyramids. Hexa-Core allows for reduction in number of elements for quicker solver run time and better convergence.

2.3 The Structure of Ansys CFX

The Structure of ANSYS CFX ANSYS CFX consists of four software modules that take a geometry and mesh and pass the information required to perform a CFD analysis

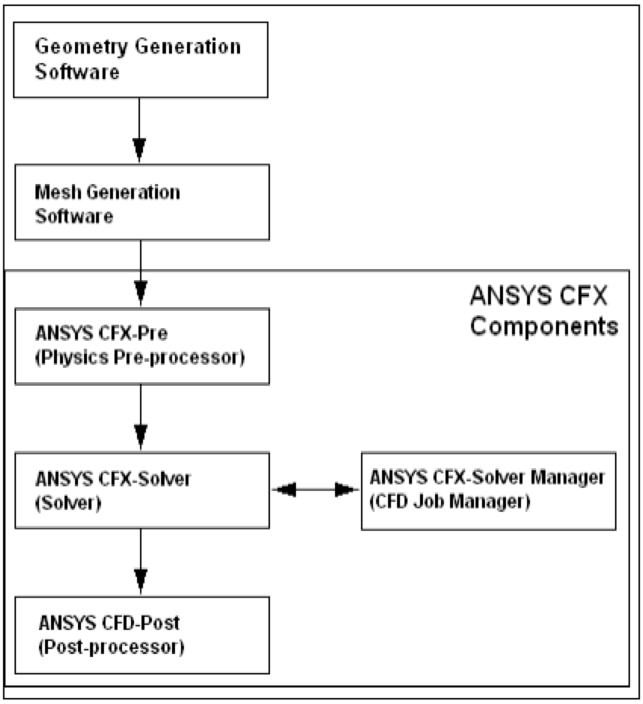

Fig 1: Flow Chart of the Project.

3. METHODOLOGY

CFD can be used to determine the performance of a component at the design stage, or it can be used to analyze difficulties with an existing component and lead to its improved design. The process of performing a single CFD simulation is split into four components:

1. Creating the Geometry/Mesh
2. Defining the Physics of the Model
3. Solving the CFD Problem
4. Visualizing the Results in the Post-processor

3.1 Creating the Geometry/Mesh

This interactive process is the first pre-processing stage. The objective is to produce a mesh for input to the physics pre-processor. Before a mesh can be produced, a closed geometric solid is required. The geometry and mesh can be created in the Meshing application or any of the other geometry/mesh creation tools.

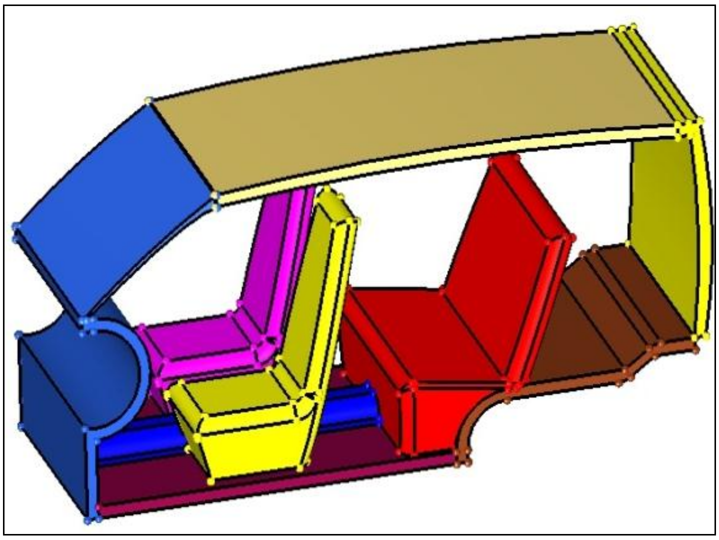

Fig 2: Geomtery of Cab Model.

3.2 Cad Editing

These are the following steps that have been followed to create Cooling vents.

1. Import the *.stp geometry to ANSYS ICEM CFD
2. Cleanup the geometry
3. Create first set of vent (first possible iteration)

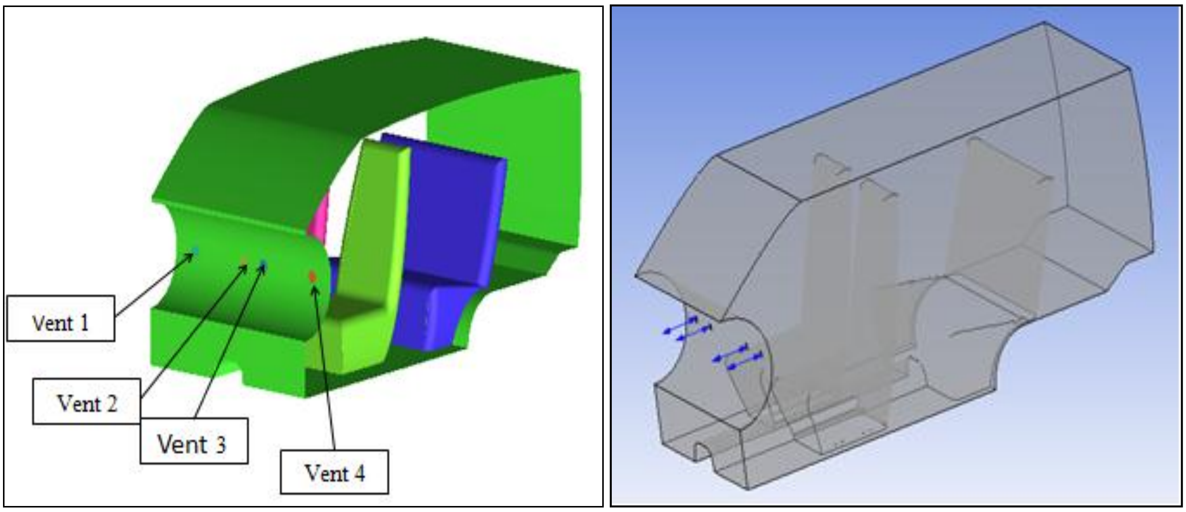

Fig 3: Model after importing to CFX Pre.

4. RESULT AND DISCUSSION

Temperature and velocity distribution of the air flow across the planes that are arranged across different vents. Two iterations are performed. Firstly, vents are kept in a row. Secondly, vents 2 and vents 3 are moved by 20^0.

5.1 First Iteration

In the first iteration all the vents are kept in a row and the planes at each vents are shown in the below figure.

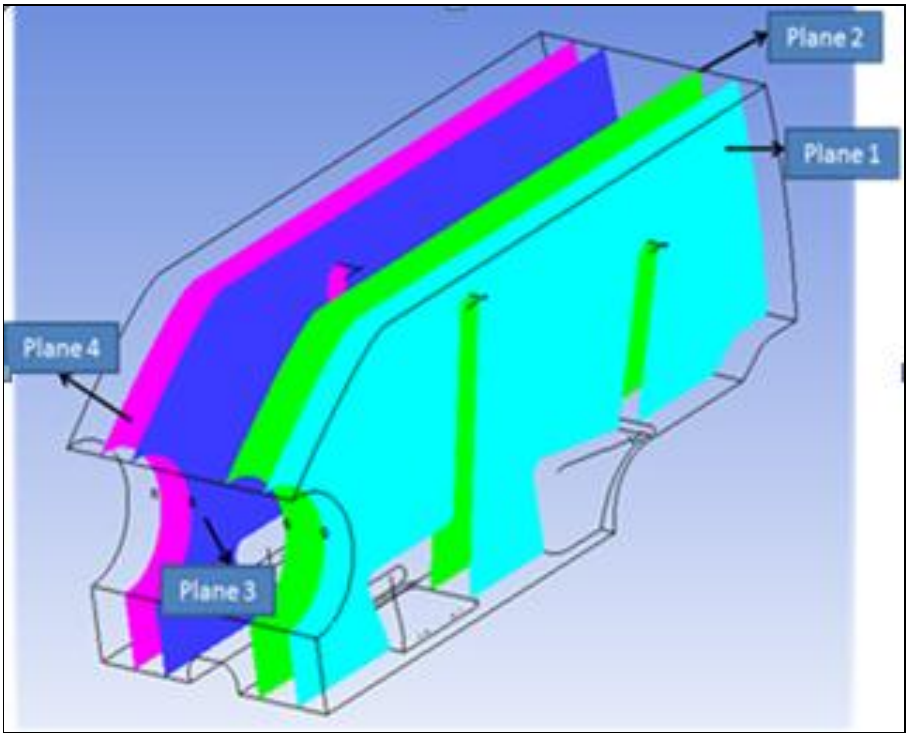

Fig 4: Post-Processing: Rating Planes at each Vent Location.

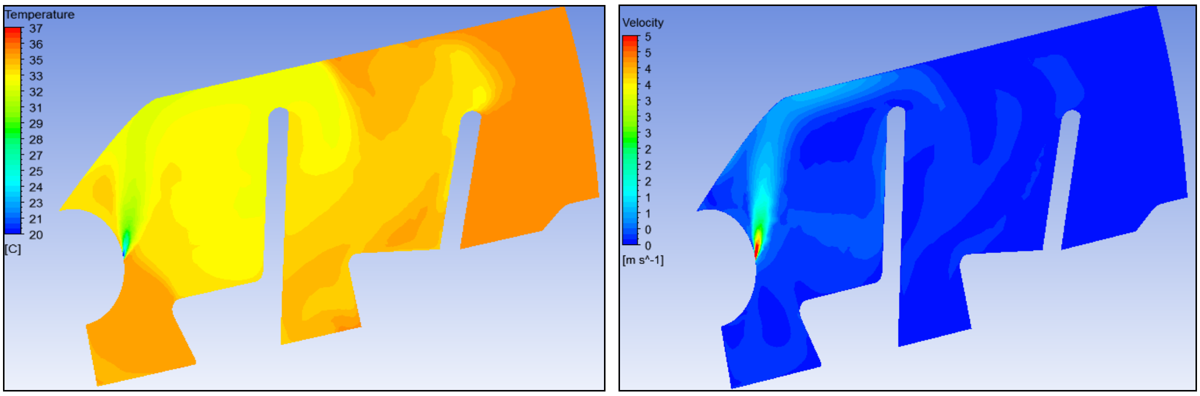

Fig 5: Temperature & Velocity Contours at Rating Plane 1 for Iteration 1.

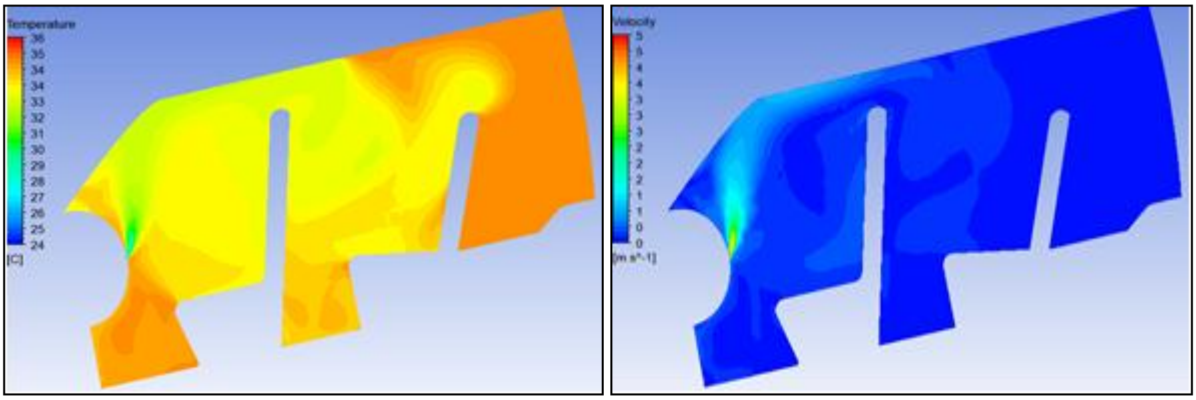

Fig 6: Temperature & Velocity Contours at Rating Plane 2 for Iteration 1.

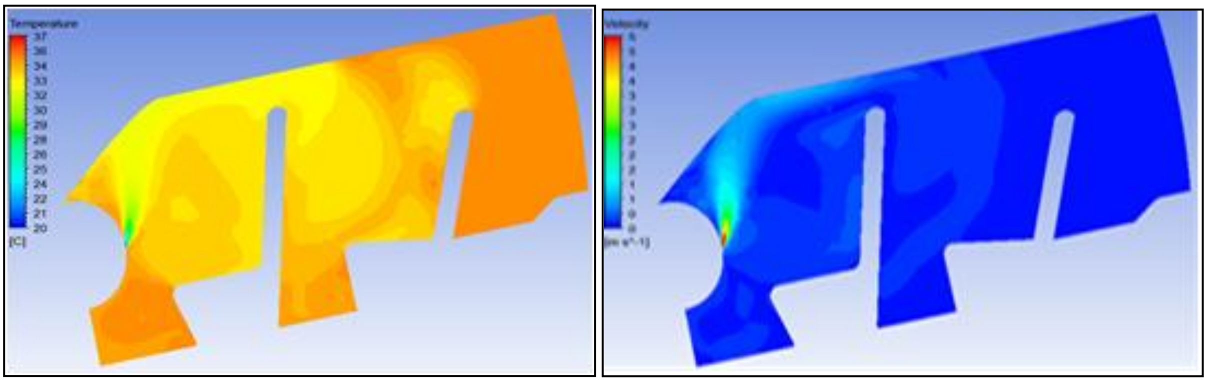

Fig 7: Temperature & Velocity Contours at Rating Plane 3 for Iteration 1.

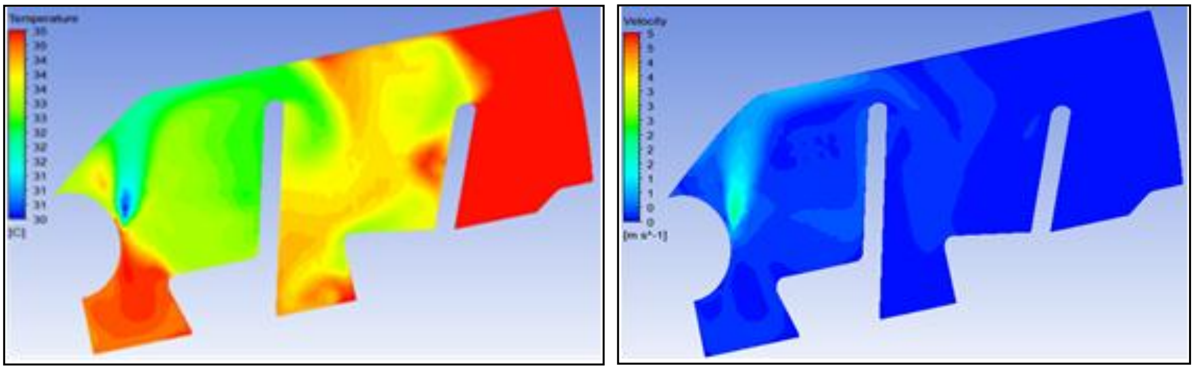

Fig 8: Temperature & Velocity Contours at Rating Plane 4 for Iteration 1.

5.2 Second Iteration

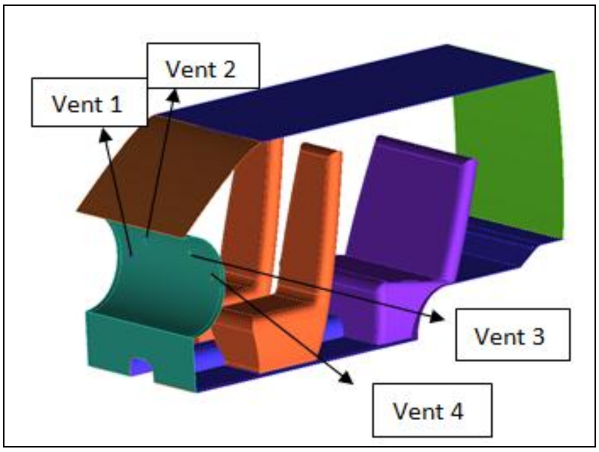

Fig 9: Vents 2 and 3 Moved by 20^0.

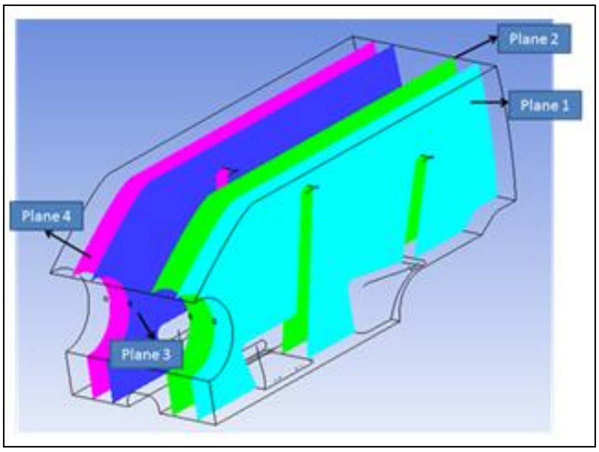

Fig 10: Rating Planes at each Vent Location.

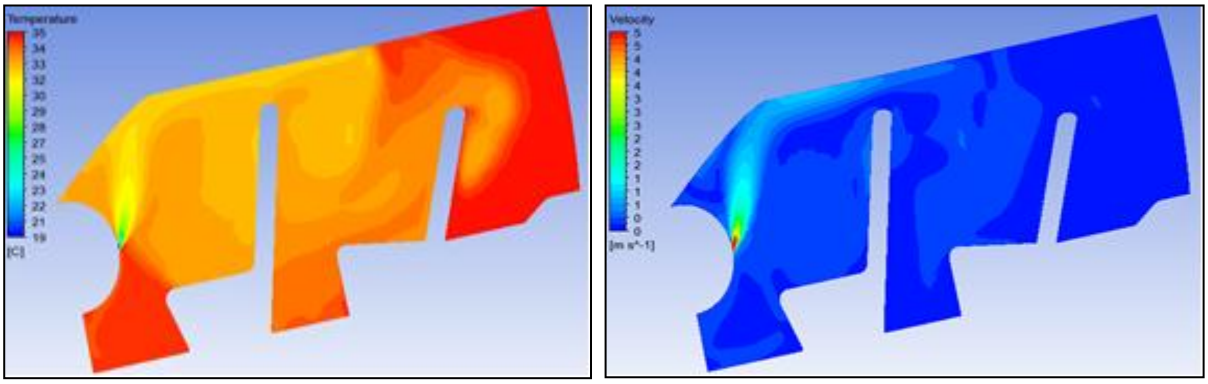

Fig 11: Temperature and Velocity Contours at Rating Pane 1 for Iteration 2.

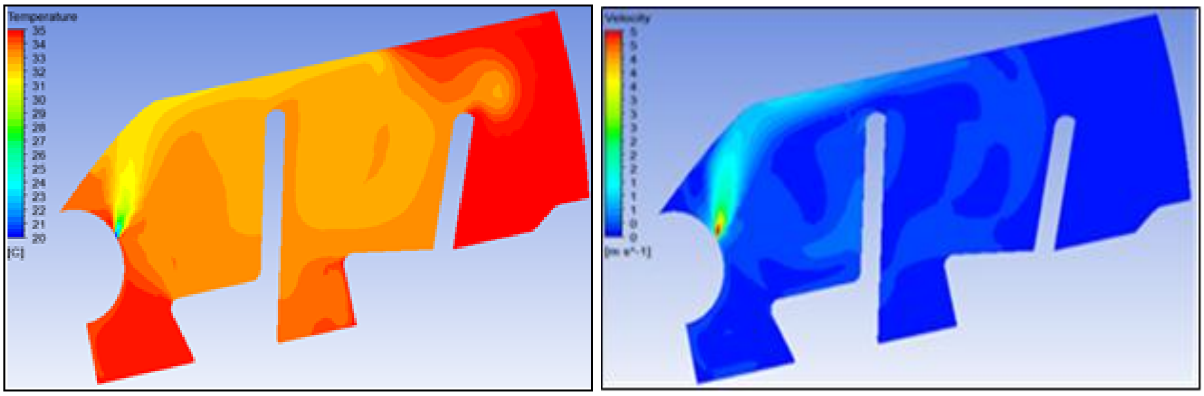

Fig 12: Temperature and Velocity Contours at Rating Plane 2 for Iteration 2.

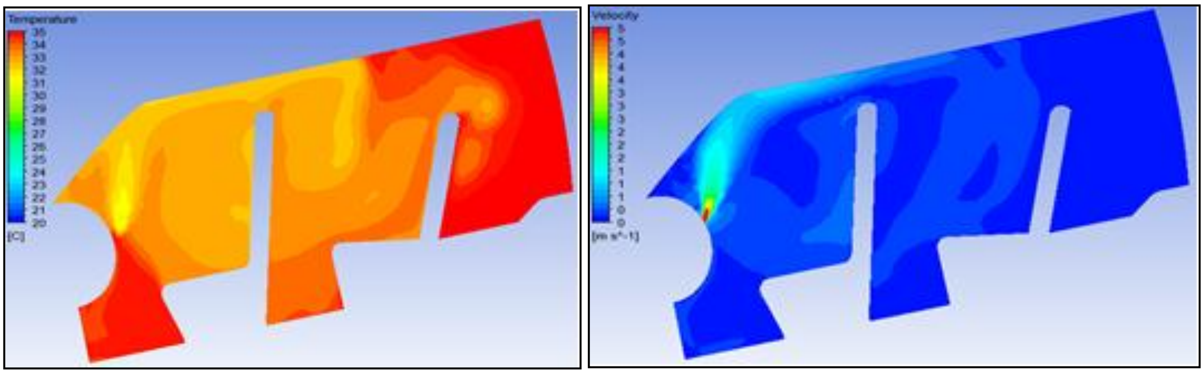

Fig 13: Temperature and Velocity Contours at Rating Plane 3 for Iteration 2.

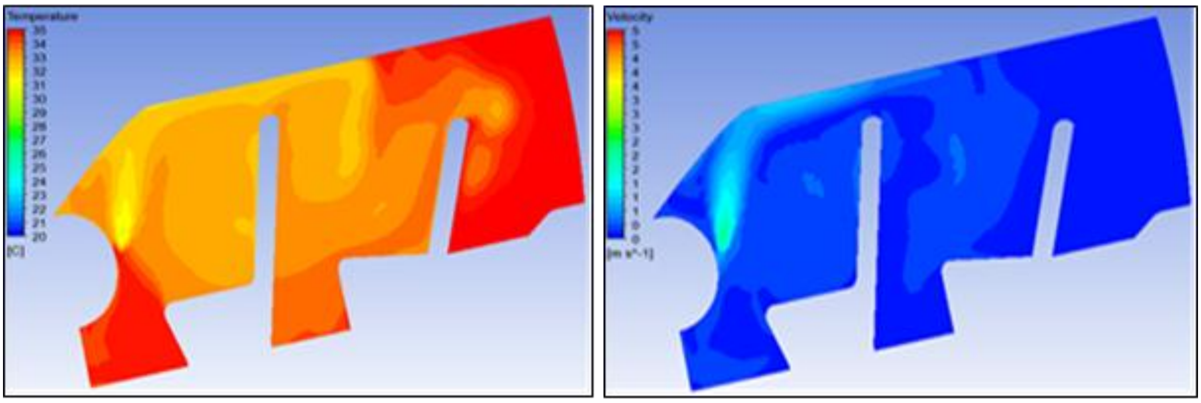

Fig 14: Temperature and Velocity Contours at Rating Plane 4 for Iteration 2.

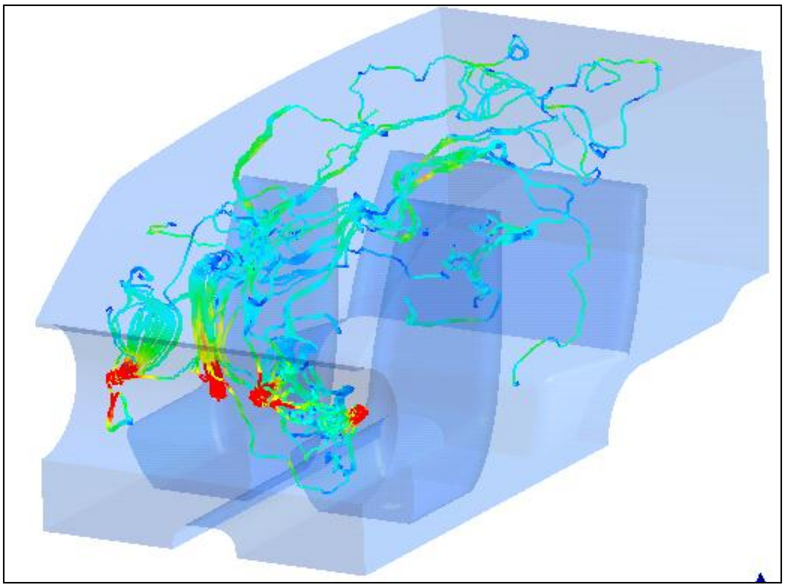

Fig 15: Isometric View when all Vents Open.

5. CONCLUSION

The efficiency of the cooling inside the cabin depends mainly on the location and shape of the cooling vent. It is reported that the location of the vent plays major role compare to shape. In the all above iteration show how the temperature varying across the cabin because of change in the cooling vent location.

Iteration 2 (vent 2 & 3 moved 20° up) shows the better distribution of temperature across the cabin at all the rating planes as shown in the above given fig.

REFERENCES

1. Hussain H.Al-Kayeim , M.Firdaus Bin M.Sidlik, Yuganthira R.A.L Munusammy, "*Study on the Thermal Accumulation and Distribution Inside a Parked Car Cabin*", American Journal of Applied Sciences, 7(2010) 784-789
2. Alex Alexandrov,Vladimir Kudriavtsev,Marcelo Reggio, "*Analysis of Flow Patterns And Heat Transfer in Generic Passenger Car Mini-Environment*", 9th Annual conference of the CFD Society of Canada, 2001. Jalal M.Jalil, Haider Qassim Alwan, CFD Simulation for a Road Vehicle Cabin, JKAU: Eng Sci, Vol.18 no.2, 123-142, 2007.
3. Huajun Zhang, Lan Dai, Guoquan Xu, Yong Li, Wei Chen, Wen-Quan Tao, "*Studies of air-flow and temperature fields inside a passenger compartment for improving thermal comfort and saving energy*", Applied Thermal Engineering 29(2009) 2022-2027