

Energy Efficient HVAC System with Spot Cooling in an Automobile - Design and CFD Analysis

Debashis Ghosh and Mingyu Wang
Delphi Automotive Systems

Edward Wolfe
Delphi Thermal Systems

Kuo-huey Chen, Shailendra Kaushik and Taeyoung Han
General Motors Company

ABSTRACT

Spot, or distributed, cooling and heating is an energy efficient way of delivering comfort to an occupant in the car. This paper describes an approach to distributed cooling in the vehicle. A two passenger CFD model of an SUV cabin was developed to obtain the solar and convective thermal loads on the vehicle, characterize the interior thermal environment and accurately evaluate the fluid-thermal environment around the occupants. The present paper focuses on the design and CFD analysis of the energy efficient HVAC system with spot cooling. The CFD model was validated with wind tunnel data for its overall accuracy. A baseline system with conventional HVAC air was first analyzed at mid and high ambient conditions. The airflow and cooling delivered to the driver and the passenger was calculated. Subsequently, spot cooling was analyzed in conjunction with a much lower conventional HVAC airflow. Spot cooling was achieved by strategically placing multiple nozzles in the vehicle directed at specific body parts. Nozzle design and nozzle locations were paramount to the success of comfort delivery and achieving energy efficiency through spot cooling. CFD analysis was mostly done in steady state mode for designing the spot cooling system. Based on the results of CFD simulation and heat transfer analysis, spot cooling airflow quantities and temperatures were recommended for implementation in the vehicle and testing in the wind tunnel. Lower cooling requirement on the conventional HVAC system due to spot cooling is the primary basis for energy savings achieved in AC mode. On a pure heat transfer basis, significant improvement in cooling delivery to the occupant was achieved through a quad combination strategy of spot cooling at significantly lower airflow and cooling assist from the conventional HVAC system.

CITATION: Ghosh, D., Wang, M., Wolfe, E., Chen, K. et al., "Energy Efficient HVAC System with Spot Cooling in an Automobile - Design and CFD Analysis," *SAE Int. J. Passeng. Cars - Mech. Syst.* 5(2):2012, doi:10.4271/2012-01-0641.

1.0. Introduction

Air conditioning systems contribute to a significant part of vehicle fuel mileage. Multiple pull factors to improve the fuel efficiency of automotive systems calls forth different approaches to significantly improve the current HVAC systems. From the perspective of vehicle exterior, glazing, shading from the sun, reducing vehicle thermal mass, paint improvements, etc.; have been investigated to reduce the solar load into the cabin. On the vehicle interior side, spot cooling is one of the approaches that can enable energy savings in vehicle air-conditioning without sacrificing comfort. The present study focuses on devising effective spot

cooling strategies that aim to reduce the energy consumption by the traditional A/C systems.

As opposed to buildings that can generally maintain a homogeneous comfortable environment for the occupants, the fluid-thermal environment inside the car is very non-uniform. Thermal non-homogeneity is attributed to angular solar loads, variation in thermal mass, material properties, occupant metabolic and physiological variation etc. The inhomogeneity of the thermal environment directly cause comfort variation to the passengers. An additional complicating factor is that even under a uniform thermal environment, the subjective perception of thermal comfort varies from subject to subject. Thus the matter of thermal comfort in a non-uniform

environment is quite complex and has been a subject of its own in the scientific community. Arens et al. [1-2] is among some of the early studies on thermal sensation and comfort in vehicle environment. In this study, the body core and skin temperatures were measured using injected radio pills and cabin temperatures were measured with externally attached thermocouples while the passengers took the comfort rides in an environmental tunnel. Kaushik et al. [3] describes one of the early analysis and simulation work on cooling based on the recommendations of University of Berkeley comfort study. The scope of the study, however, is limited to more uniform tent like environment.

The present work provides detailed analysis of the impact of spot cooling using CFD as an analysis tool and attempt to propose effective spot cooling strategies that can achieve satisfactory comfort for ninety percentile of the population.

1.1. Spot Cooling

One of the limitations in delivery of cooling to the occupants using current HVAC systems is that a significant portion of compressor work is expended in cooling the large thermal mass of the vehicle. The cooling requirement of a passenger is about 1-1.5Met (100-150watts). The total amount of energy spent in cooling a 1-1.5 Met individual is disproportionately large. The inefficient expenditure of cooling energy is largely attributed to the following: A focal source of air delivered as a diffuse stream distant from the occupant from large HVAC vents causes the airflow to not reach the occupant at optimum desired velocity for comfort; Large thermal mass of the cabin and other parasitic heat loads cause the cabin to cool down at suboptimal rate; Little or no ability to rapidly cool or heat the most uncomfortable body part upon entering a soaked vehicle cabin.

In the spot cooling or heating approach, airflow is delivered in a distributed fashion. By going to distributed airflow delivery much higher air velocities are achieved around the occupant. Delivery of cooling/heating is much more direct and instantaneous on the body part of significance. In HVAC systems designed with spot cooling, it is possible to efficiently shift the focus of comfort delivery from maintaining a comfortable cabin to maintaining a comfortable occupant. In the process of delivering comfort to the occupant quicker, the cabin cool down and warm up rates are partially compromised for the purpose of achieving energy savings. Running a warm cabin but achieving comfort through spot cooling forms the essential basis of realizing energy savings. Thus the passengers can potentially achieve much quicker time to comfort while the in-car conditions are still warm or cold.

Ideally, with spot cooling, the airflow is conditioned to the most appropriate temperatures customized by the occupant. So the chance of overcooling a body part, as is common in conventional HVAC systems, is minimized. Maximum benefit reaped with spot cooling is that with distributed cooling delivery it is possible to achieve a comfortable, true micro-climate environment around the

occupant whereby the occupant is somewhat decoupled from the non-homogeneous, stratified, anisotropic, uncomfortable cabin. Design of nozzles, duct layout, tempering of spot cooling air are the key enablers for the success of spot cooling.

In the present paper we describe an approach to augment the conventional HVAC system with spot cooling technology for a five seater crossover vehicle. The work is sponsored by the Department of Energy (DOE) under the joint stewardship of Delphi Thermal Systems, General Motors (GM), and the University of Berkeley (UCB). As a primary goal of the project the energy saving for the vehicle shall be realized by running the vehicle at elevated in-car conditions while maintaining equivalent occupant comfort with spot cooling. Also per the goal of the DOE project, tempered air for spot cooling shall be supplied by conditioning the in-car cabin air by thermoelectric devices.

The description in this paper is divided into five segments.

1. The first segment describes developing the cabin CFD model.
2. The second segment describes the efficacy of individual local cooling strategies in the vehicle based on identification of sensitive body parts.
3. Based on the efficiency of individual spot cooling strategies, a couple of potential combination cooling strategies were proposed for summer tunnel tests, which are described in the third segment.
4. The fourth segment compares the efficacy of HVAC assist spot cooling strategy set at an elevated cabin temperature with a conventional HVAC system set at a lower cabin temperature.
5. The last segment describes the comparison of CFD prediction with tunnel test data.

2.0. Developing the Cabin CFD model

The CAD for the crossover vehicle was generated in Unigraphics. Only the shell of the vehicle was used for meshing, including the doors and windows. The mass effects and thickness for the respective surfaces were specified in the boundary condition using shell conduction. The meshing was done in ANSYS design modeler. FLUENT was used as the solver. To model spot cooling accurately the mesh resolution needed to be high for all the nozzles and flow from the nozzles. The model also needed to comprehend accurately all the impingement and stagnation heat transfer from the occupant. Very important jet entrainment dynamics had to be captured by the simulation. With the cabin operating at elevated temperatures, the small amount of spot cooling air discharged at temperatures significantly different from the cabin conditions rapidly lost temperature in cool and warm environment. The higher the thermal differential between the cabin and spot cooling air, the greater is the adverse

temperature pick up due to entrainment. The higher the airflow rate, the stronger is the jet ingestion dynamics. All these effects needed to be accurately captured by CFD. All the analysis was run in steady state. To keep the overall mesh size manageable, a domain decomposed meshing was used for all the nozzles. A fifteen to twenty degree conical/ellipsoidal solid for the flows emanating from each nozzle was created in the middle of the cabin for meshing spot cooling. The seats were modeled as a hollowed out surface, in the process actual thermal mass of the seat was neglected. However, an approximate effect of seat thermal mass was accounted for in the CFD model through seat shell conduction with wall thickness of 10mm. For doing the basic design work the seat boundary conditions were specified as adiabatic wall. However, more realistic boundary condition with thermal mass effects for a solid foam seat could also be modeled if more accuracy is desired.

Two fifty percentile male dummies from the Delphi mannequin library for cabin work were incorporated as front passengers in the CFD model. The body surface mesh sizes for the dummies were around 5 mm. The dummy mesh is very important when trying to ascertain the efficacy of each local cooling strategy. A 22 segment mesh was developed for each dummy for comfort modeling work in RADTHERM. However, the UCB comfort model could only handle 16 body segments. So some of the velocity and temperature gradients induced by spot cooling got smoothened in mapping the fluid-thermal data from 22 body segment in FLUENT to 16 body segment in RADTHERM. For rank-ordering different design option which was primarily done based on temperature and velocity field around the occupant, 22 body segment was useful, especially for the neck, face, head and chest.

The total model size was around 5 million cells. Convection heat transfer coefficient at 30 mph and far-field temperature boundary condition was imposed for the entire exterior wall, activated with shell conduction. The heat transfer coefficients were different for the windshield and the roof walls compared to the other vertical surfaces. Discrete Ordinate radiation model was used for radiative heat transfer from the walls. Solar load was imposed per the NREL solar calculator implemented in FLUENT. Engine heat, exhaust system heat transfer into the cabin was neglected.

The windows and windshield were imposed with green-glass material properties. For most of the design work the flow from the nozzles were treated as in outside air mode. Velocity/mass flow inlet boundary conditions were specified for the nozzles. More details will be provided in the CFD validation section.

3.0. Design of Single Spot Cooling Nozzles- Location and Airflow

The main objective of this design effort is to obtain the best nozzle locations and airflow from the nozzle. The following sets of nozzles for each passenger described in the

adjoining table were investigated in the CFD analysis during the design phase of the project. UCB recommendations were incorporated for cooling of the targeted body part per the findings of UCB pilot study in a more uniform tent enclosure

Preferably, two sets of nozzles were analyzed for the initial design. The nozzles were kept symmetric with respect to the occupant, especially for the exposed body parts like face and neck. People do not like asymmetric (left-to-right) temperature and velocity differentials, nor do they tolerate large thermal gradients on the body. It is incumbent upon the HVAC system designer that with spot cooling we maintain the skin temperature within a narrow range for different ambient conditions. Small change in skin temperature can evoke a large change in comfort response. For this reason the design of spot cooling enabled HVAC system should be robust enough not to significantly overcool or overheat a particular body part. Instead it should maintain the body surface temperatures of the occupant within a narrow range. With spot cooling, occupant comfort is achieved by delivering the cooling through a combination of higher heat transfer coefficients on the targeted body part and controlled air temperature delivered to the body.

Also two nozzles helped with air side pressure drop and made the design more robust to passenger subjectivity and mounting variations. A couple of nozzle diameters were analyzed for each body part. For the final design a single effective nozzle diameter was used for spot cooling of each body part. The nozzle diameters were smaller for the face and hands. The chest cooling nozzles had the biggest diameter. The lap cooler had a high aspect ratio slotted exit.

The location and directivity of the nozzles were key to the success of spot cooling. The design considerations that were taken into account for spot cooling were the following: a) Local control had to be provided to the occupant to direct the nozzle towards and away from the body; b) Each spot cooling technology had to work for ten percentile to ninety percentile of the population. To be effective in delivering cooling the nozzles had to be close to the passenger. Due to occupant size and seat position variation, airflow spread on the body dictated that a minimum nozzle standoff distance be maintained from the occupant. Finally, in an automotive environment, spot cooling is designed to work in conjunction with a conventional HVAC system. Unlike in buildings or in airplanes, where the occupant surrounding is relatively large and fluid-thermally quiescent, the vehicle cabin volume is small compared to the occupant, and the convection velocities are higher than in buildings. It is therefore important to comprehend in the design the strong interaction between the nozzles, HVAC air and occupants. CFD analysis was performed to locate the nozzles for cooling of each targeted body part. Simple conical nozzles were used for phase-I of the study. The nozzle diameters were determined by air exit velocity considerations. The nozzle locations and nozzle velocities were dictated by the cabin air entrainment dynamics, airflow spread, the circulation pattern induced by the nozzle flow and the range of body surface impingement

Nozzle Description	Location	Target
2 Face Cooling Front nozzles	Headliner	Face
2 Face Cooling Rear nozzles	Seat	Face
2 Chest Cooling Nozzles	A Pillar & Roof Console	Chest
1 Lap Cooler	Dashboard	Thigh, Arm, Pelvis
		Hands & Face
2 Hand Cooling Nozzles	Steering Column	Hands

velocities well tolerated by the occupant. CFD analyses were very useful in optimizing the nozzle location and evaluate the sensitivity of nozzle directivity on cooling performance.

3.1. FACE COOLING

Airflow from the front face cooling nozzles was directed towards the lower face of the passenger. Two different options were investigated for face cooling a) nozzles located in the front on the headliner and b) nozzles located in the rear in the seat head rest.

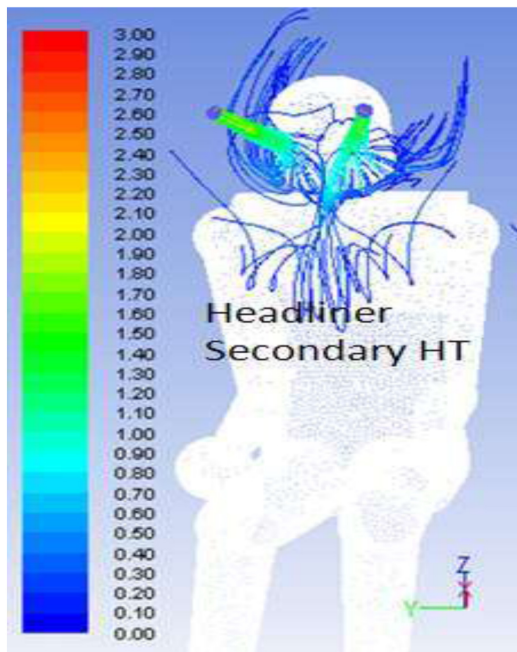


Fig 1. Airflow path from two front face cooling nozzles on headliner

Fig 1 shows two front headliner face cooling nozzles pointing downwards onto the face. Fig 2 shows the results of face cooling from the rear from two side mount nozzles. Airflow was directed sideways onto the face from the seat

nozzles. The front face cooling nozzles from the headliner showed better potential for spot cooling as opposed to the side or rear mount face nozzles.

As evident in Fig.1, the airflow being directed downwards onto the face had longer time to interact with the passenger. Thus an additional chance for secondary heat transfer on the chest region after primary cooling of the face was achieved. At low airflows the air stream directed downwards does not have a significant chance of causing dry eyes due to the flow up wash.

In all CFD runs the face airflow was chosen such that air velocity on the face was not high. It is very important for face cooling nozzles not to have high velocities impinging on sensitive and exposed part of the body.

3.2. CHEST COOLING

Chest cooling nozzles for the front passengers could be mounted in two ways a) symmetrically with respect to the occupants on the headliner and roof console, and b) asymmetrically both on the A pillar or cabin frame. Fig 3 shows the velocity contour and flow path for two chest cooling nozzles mounted on A pillar.

It demonstrates the airflow effectiveness of the 2 front nozzles mounted on A pillar. Even at low airflows from the chest cooling nozzles mounted on the A pillar, the airflow spread over the body is very good. The air wraps around the chest and then flows on to the abdominal area. By properly directing the nozzle towards the chest, it is possible to achieve additional secondary heat transfer around the sweat regions in the armpit areas which provides enhanced comfort response attributed to evaporative cooling. Fig 4 shows the airflow pathlines from chest cooling nozzles mounted on A pillar.

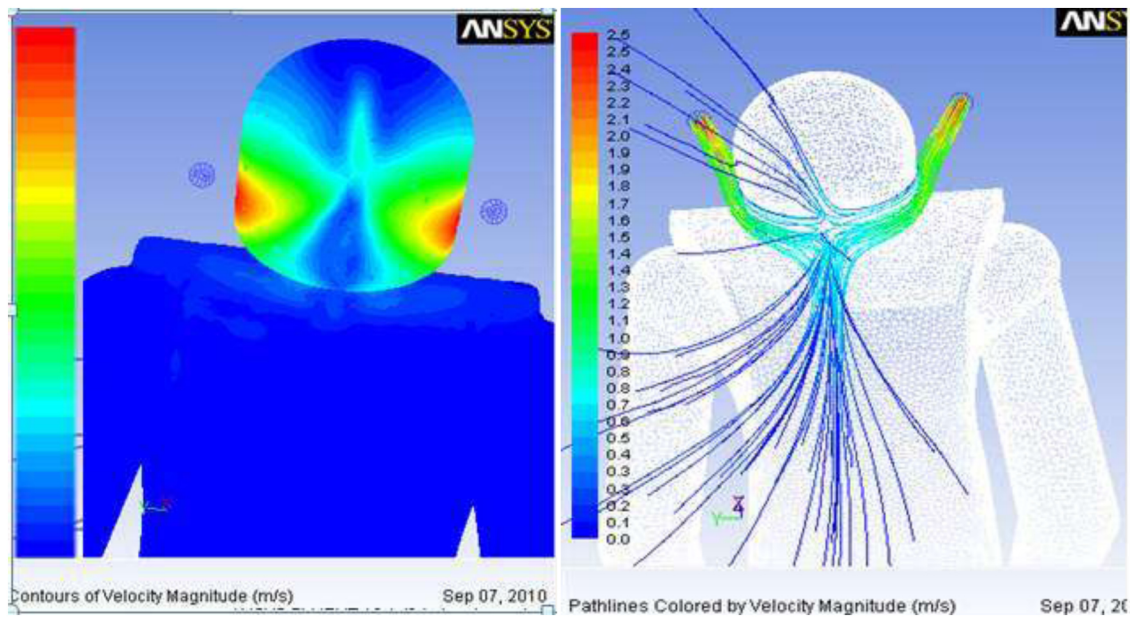


Fig 2. Airflow path and velocity contour from 2 rear face cooling nozzles mounted on seat

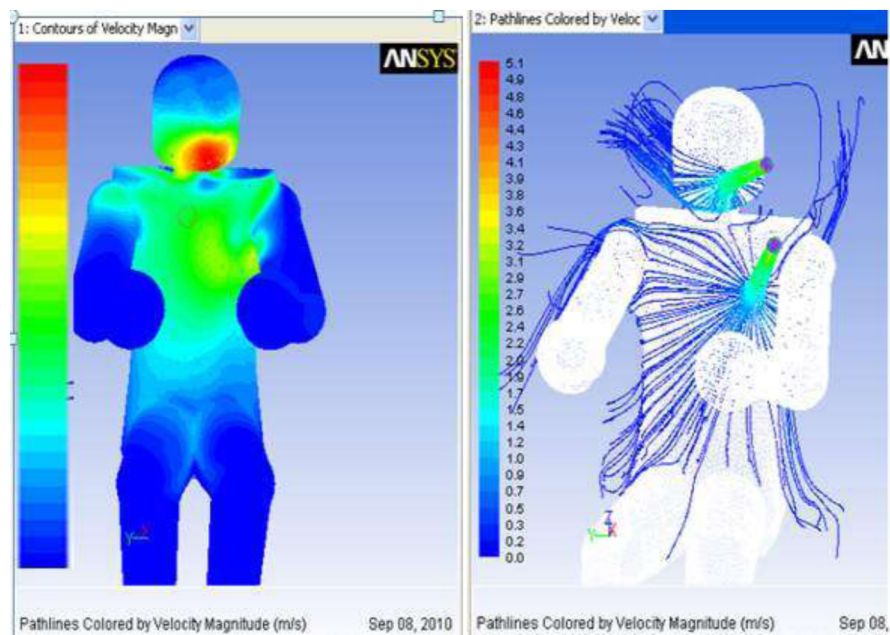


Fig 3. shows airflow path & velocity contour from two A pillar nozzles directed to face & chest.

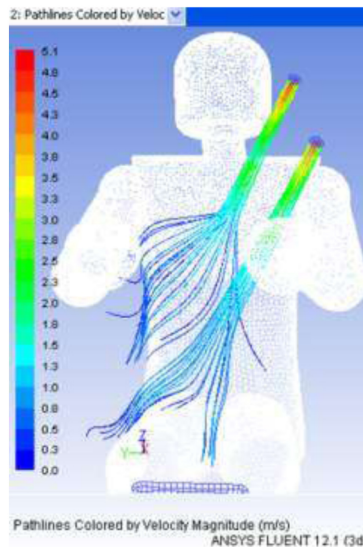


Fig 4. shows airflow path from two chest cooling nozzles in A pillar

Chest cooling nozzles are extremely effective in evoking a good comfort response. These nozzles have relatively high airflow. The higher chest cooling airflow establishes a strong recirculation pattern in the cabin disposed in front of the occupants. This recirculating flow induced by chest cooling flow achieves the added benefit of partial face cooling. This reduces the airflow requirement from the face cooling nozzles.

A critical aspect in terms of directivity of chest cooling nozzles should be such that airflow be not very high around the exposed body part and direct impingement of chest cooling airflow be avoided as body surface velocities are high.

3.3. LAP COOLING

From pure heat transfer considerations the lap cooling air was found to be very effective in cooling the passenger. The lap cooler airflow allows one to convectively isolate the passenger from the warmer cabin by enveloping the person by a blanket of cool conditioned air. The airflow utilization in achieving cooling is very good. With a single lap cooler supplying tempered air and no additional nozzles it is possible to cool an occupant very effectively. The limitation of lap cooler is that it needs more airflow (almost 1.5-2 times) than chest cooling nozzles for the airflow to be effective in cooling. Fig 5 shows the airflow from the lap cooler. An important design consideration for lap cooler is that the nozzle discharge temperature be maintained within a narrow range or else over cooling can occur. Also, since the air flows over clothed body parts, the effectiveness of a lap cooler is insignificant if total airflow is low.

3.4. Seat Cooling

Seat cooling is one of the most effective ways of delivering comfort. Since the seat has a large thermal mass, it carries lot of parasitic heat load. Thus any attempt to isolate the passenger from a thermally soaked seat should evoke a

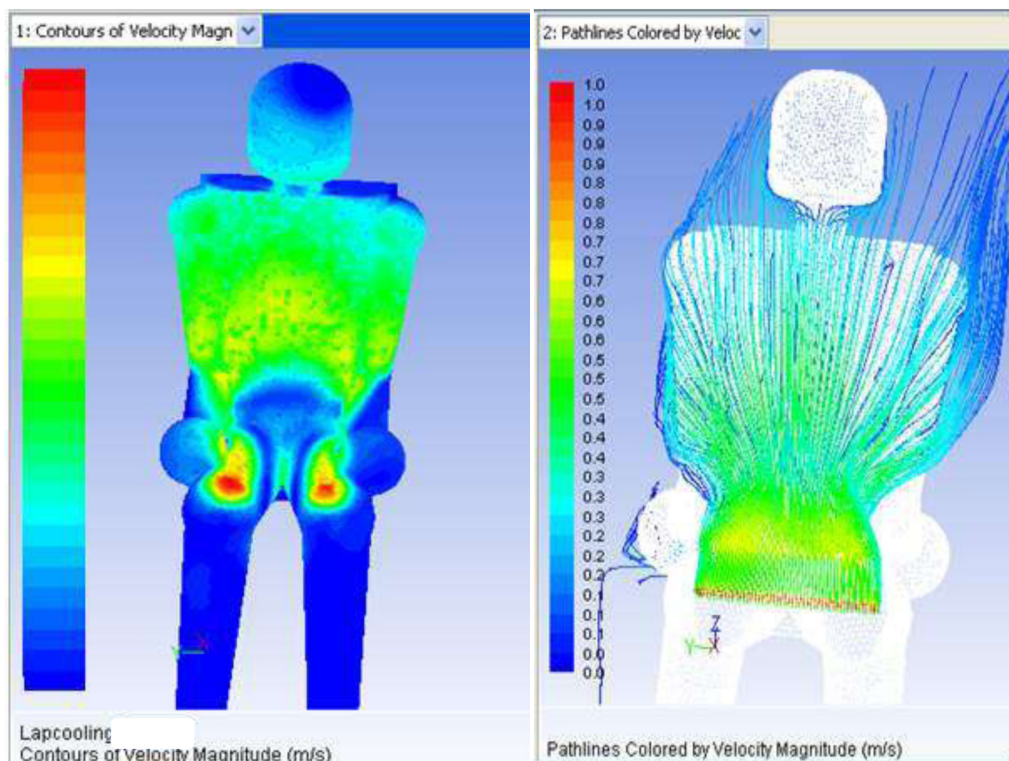


Fig 5. Airflow path and Velocity Contour from Lap cooler mounted on dashboard.