SAE International

2012-01-0641
Published 04/16/2012

doi:10.4271/2012-01-0641
saepcmech.saejournals.org

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 injested 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 use 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.

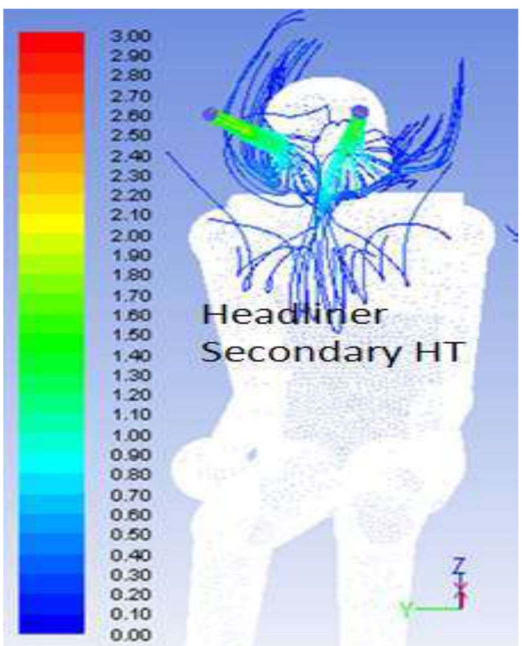

Fig 1. Airflow path from two front face cooling nozzles on headliner

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 nozles mounted on A pillar.

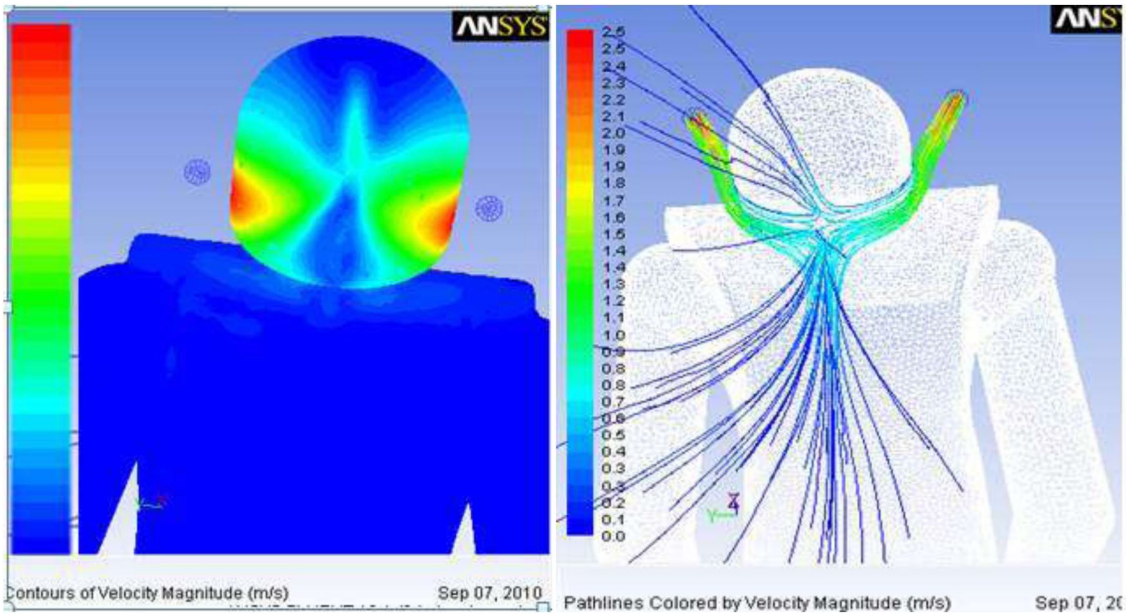

Fig 2. Airflow path and velocity contour from 2 rear face cooling nozzles mounted on seat

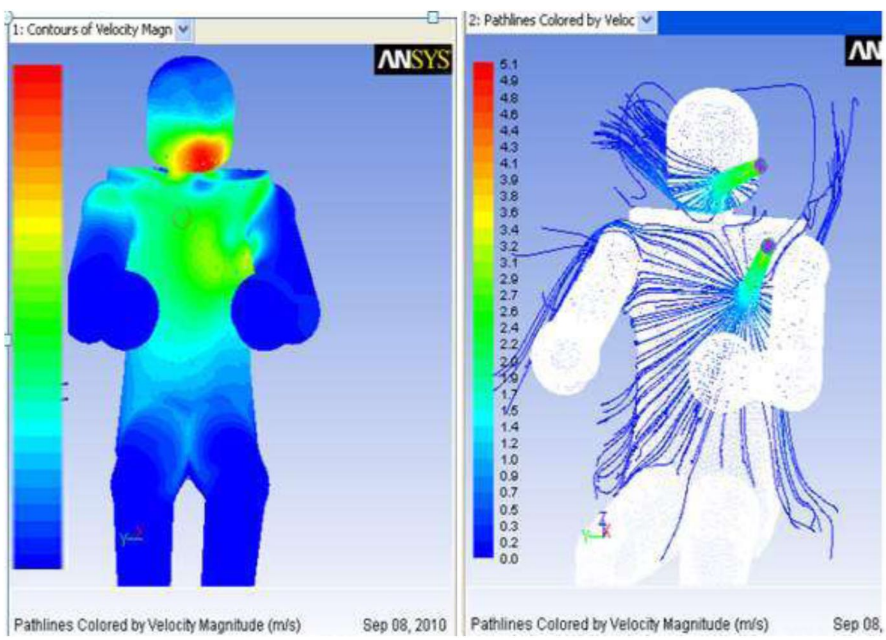

Fig 3. shows airflow path & velocity contour from two A pillar nozzles directed to face &chest.

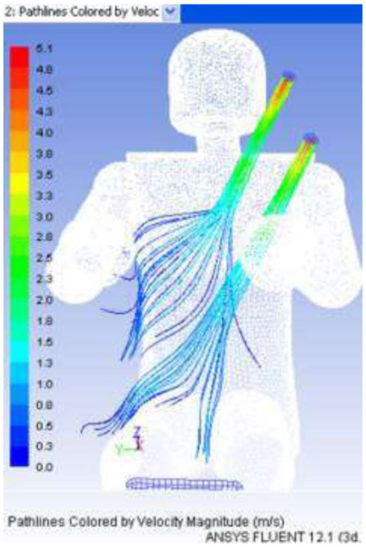

Fig 4. shows airflow path from two chest cooling nozzles in A pillar

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

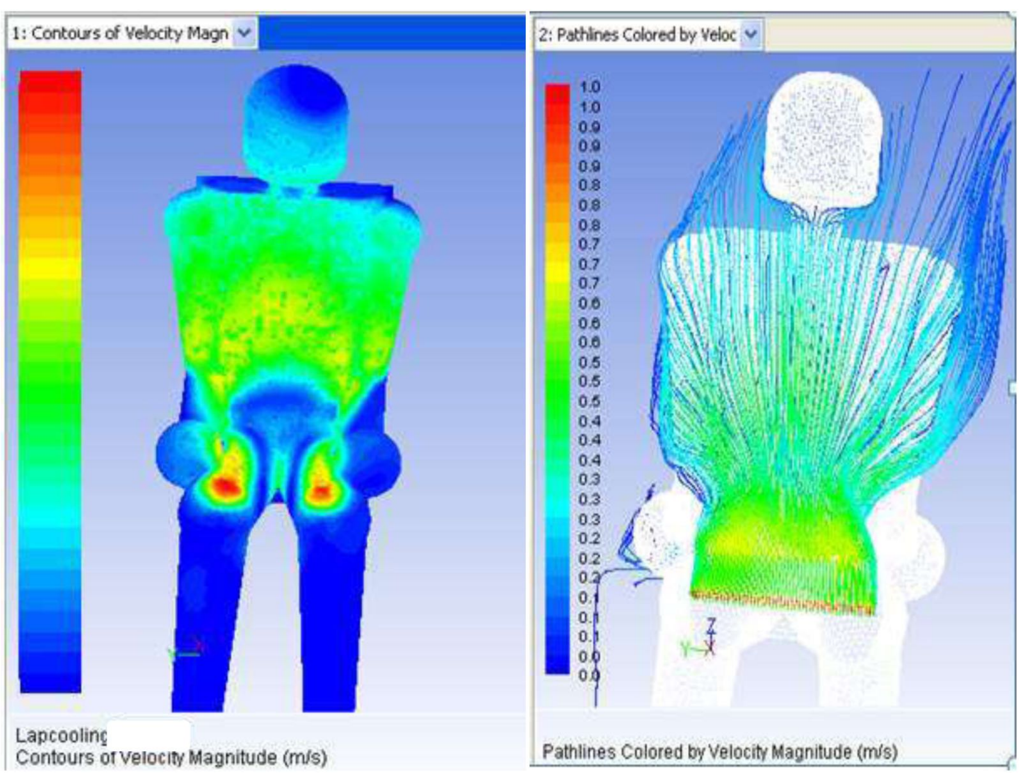

Fig 5. Airflow path and Velocity Contour from Lap cooler mounted on dashboard.

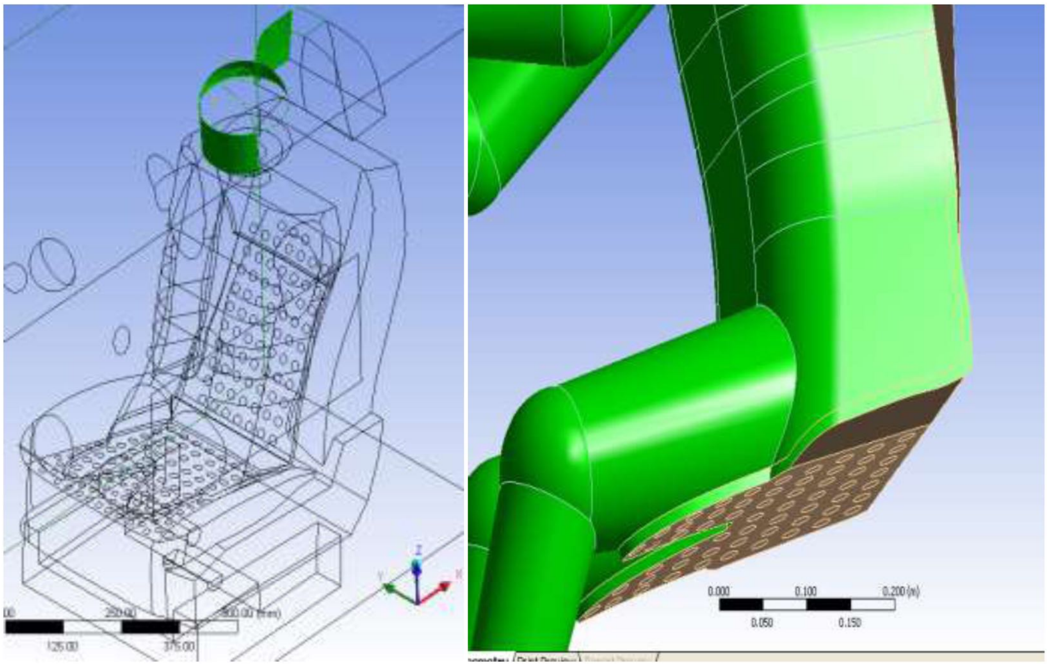

Fig 6. Ventilated seat cooling CAD model

good comfort response. Additionally, a large segment of the body - the back, the pelvis and the gluteal region comprising about greater than 30% of body surface area is in direct contact with the seat. So heating/cooling delivered through the seat is transferred directly to the occupant very effectively. Gluteal, pelvic and back cooling is achieved through both conduction and convection.

Exact simulation of a ventilated seat is very mesh expensive. Modeling flow through the small holes in the seat and finally vented into the cabin is extremely computationally expensive requiring greater 10 million cells for each seat. The other challenge is modeling contact surface heat transfer. In actuality airflow through contact surfaces occur via the interstitial passages due to curvature mis-match between the body and seat, passenger motion, etc. Dynamically some holes get opened and closed. Since airflow takes the path of least resistance more airflow shall occur through open holes, less airflow through the contact surfaces holes.

To circumvent these details we adopted an engineering approach for modeling a ventilated seat through a seat sub-model. Fig 6 shows the geometry for the seat sub model. The hole sizes were made bigger than the actual holes but still keep the same blockage factor. To reduce computer run time all contact surfaces are modeled in CFD using heat transfer coefficient, and air temperature boundary condition through clothing thickness. The heat transfer coefficient and air temperature distribution is obtained from seat submodel. Clothing conductivity values k_clothing is enhanced due to air permeation to 0.6 w/mk. Exposed holes of the seat for passenger cooling are modeled in full. Fig 7 shows a sample of velocity distribution and heat transfer coefficient variation for a ventilated seat for low airflows.

3.5. Airflow analysis of Combination Spot cooling Strategies

Based on the effectiveness of single spot cooling, combination spot cooling strategies were developed. The goal of combination cooling CFD analysis was to obtain maximum cooling efficiency with the minimum number of nozzles and total airflow. The nozzles were positioned in the vehicle to achieve maximum airflow coverage over the body of the occupant without causing large velocity and temperature gradients on the body. In identifying the best cooling combination it was important that the different streams of airflow from each nozzle not interact adversely with each other

For maximum airflow coverage at minimal overall airflow and minimum number of nozzles three distinct cooling strategies were identified for spot cooling.

Strategy 1- Face Cooling + Chest Cooling

Strategy 2- Face Cooling + Lap Cooling

Strategy 3- Face+Chest+Lap

The Figs. 8, 9, 10 illustrates the results of some of the best combination cooling strategies purely from flow efficiency and heat transfer considerations. Overall airflow coverage for both strategy 1 & 2 was very good. In Fig 9 velocity contour plot was included with 118 cfm of HVAC air and no spot cooling. HVAC air was directed towards the shoulders of the occupant resulting about 59cfm/person. With combination cooling Strategy 1 (HVAC off) with only less than one third of conventional HVAC air person for spot cooling it is

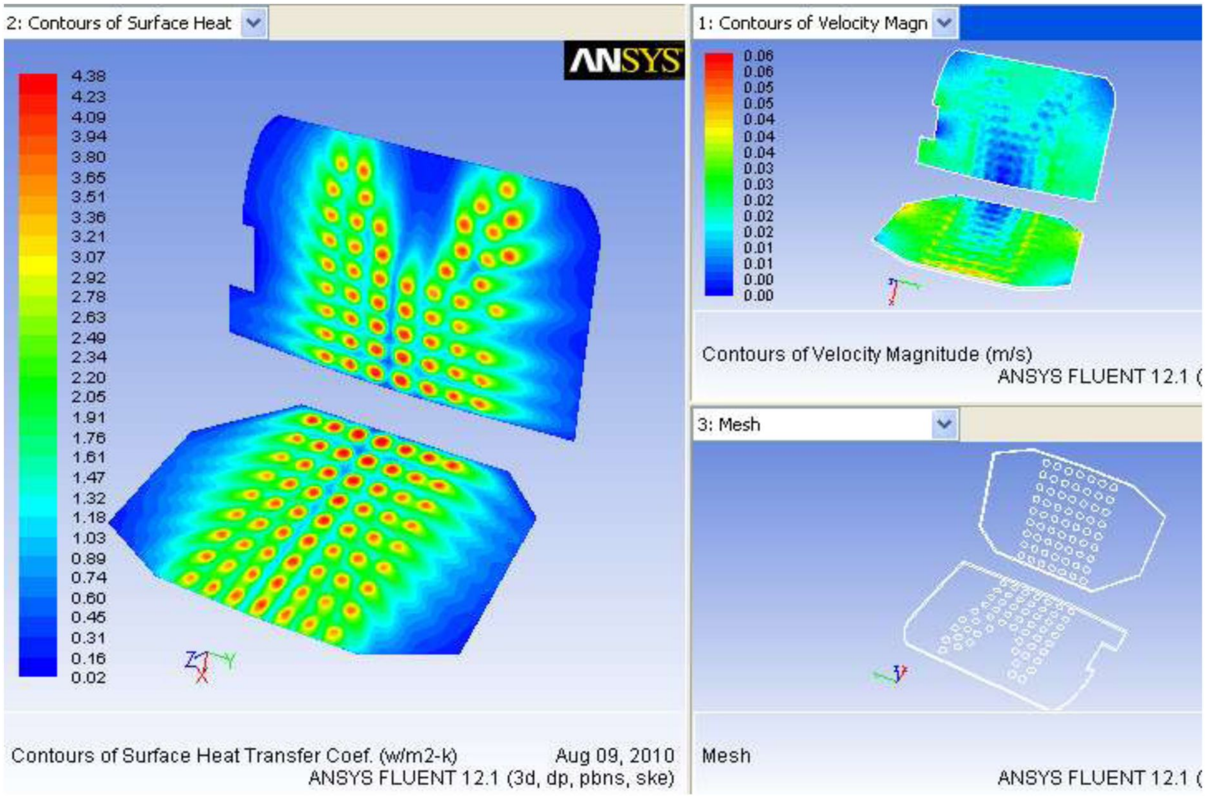

Fig 7. Velocity distribution and the convection heat transfer coefficient distributed over the seat at medium airflow.

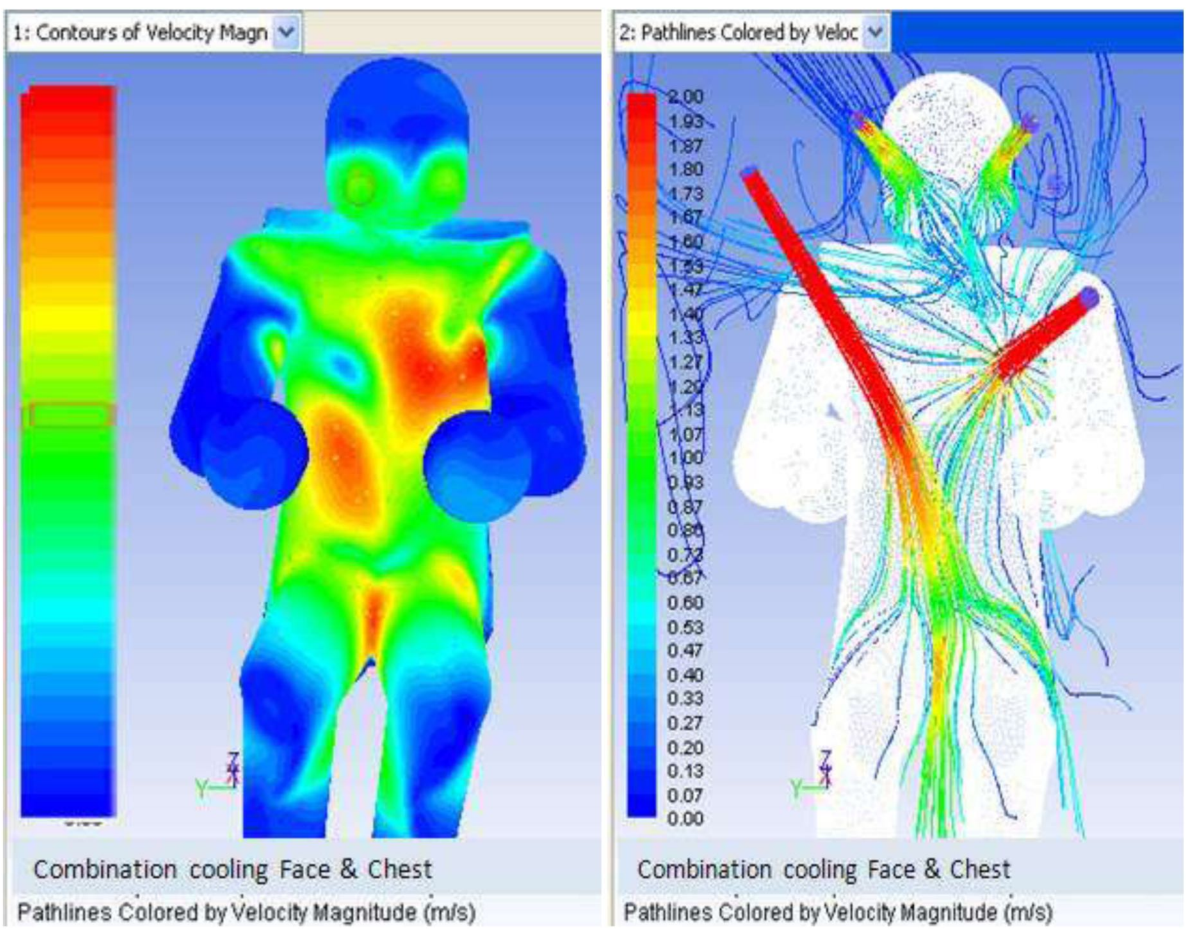

Fig 8. showing velocity field for combination cooling- Strategy1

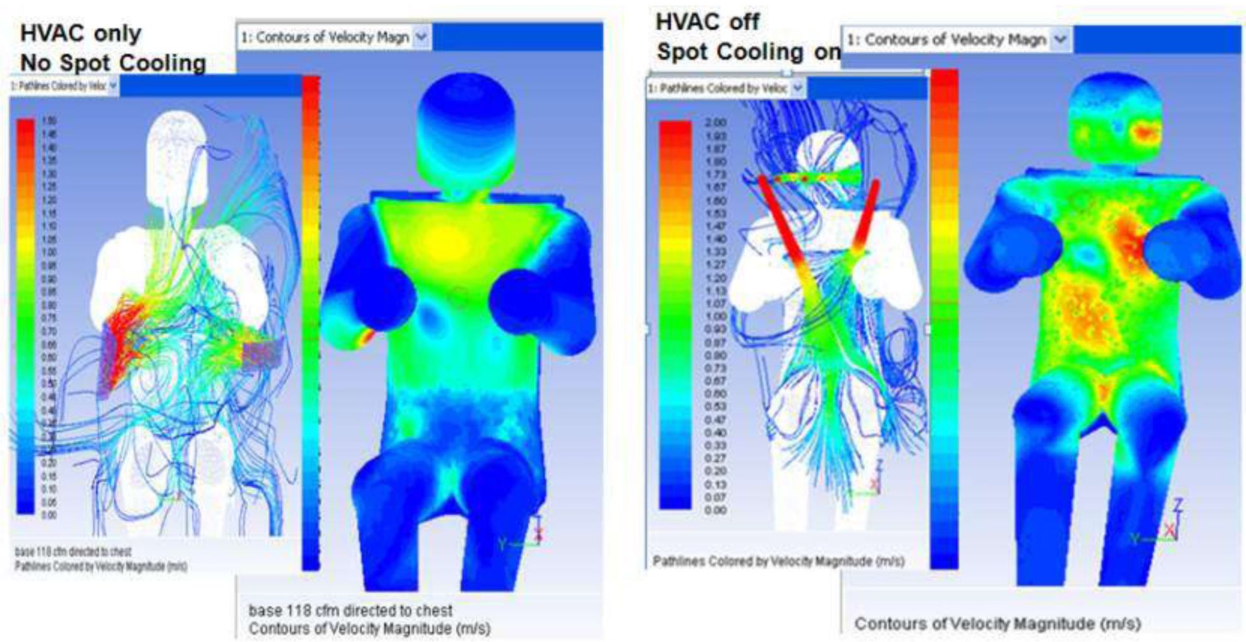

Fig 9. shows comparison of velocity field for traditional HVAC vent airflow with combination spot cooling-Strategy1 (chest-face) with only at 25% HVAC air.

possible to get around the occupant more than 50% higher velocities than conventional 60cfm/person HVAC air. With Strategy 2 (HVAC off) total spot cooling airflow of about 50% of HVAC air per person it was possible to achieve greater than 22% higher velocities than with traditional HVAC air.

The distinction between the two strategies 1 and 2 is that, Strategy 1 relies on high velocities on the clothed body part, especially around the chest, abdominal and thigh regions. Heat transfer coefficients are high, suitable for occupants who like high velocities for cooling. Per subjective comfort test Strategy 1 is more acceptable to males. In Strategy 2 cooling is achieved by enveloping the passenger by a diffuse stream of cold air. Air velocities on the body from pelvic to chest regions are lower. Women like this option per the comfort rides. The main advantage of Strategy 1 is that total airflow requirement is about 25-30% lower than Strategy 2.

Fig10 shows the flow path lines when all three modes of spot cooling were activated face, chest, and lap. Very strong interaction between the lap cooling airflow and the chest cooling flow is visible. When the HVAC is on to maintain a specific cabin temperature, the interaction effects of spot cooling air with conventional HVAC air need to be considered. Managing the adverse interaction between the different spot cooling air streams is in a finite size cabin is a very important design consideration.

Additional heat transfer calculations were done to find out the minimum amount of airflow needed to cool the driver and the passenger. It was found from pure heat transfer considerations of the occupant, that under ideal location of the nozzles and air delivery system that a little more than a third of the airflow from a conventional HVAC system was required for spot cooling in steady state. Based on the results of CFD simulation the airflow rates and discharge temperatures from the different spot cooling nozzles were specified for vehicle tunnel tests. However, analysis cannot comprehend the subjective response of airflow and temperature on each body part, so the final determination of nozzle airflow rate and best air discharge temperature relied on extensive subjective and objective testing.

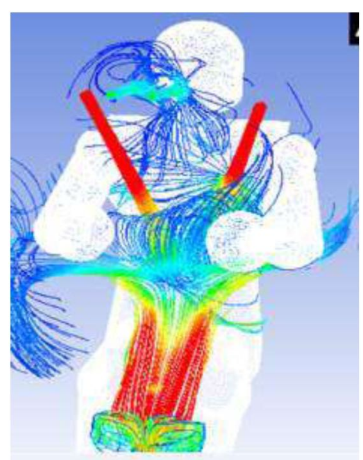

Fig 10. shows velocity pathline for tri combination spot cooling-Face, Chest and Lap

sufficient at meeting comfort at the high ambient condition, 88F/55F was a good place to start the design process. Lot more cabin data and development work is need to further the spot cooling technology at high ambient, high solar load, full passenger conditions. Henceforward, all the CFD-test validation work described below will be at 88F/55F, 500w/m^2, 30mph condition for which extensive vehicle data were collected along with the comfort rides.

4.1. BOUNDARY CONDITIONS

The inputs to the CFD model were a) Solar load into the cabin direct solar radiation 500w/m^2 incident normally and 125w/m^2 diffuse solar radiation for mid ambient conditions. b) AC vent airflow and c) AC vent temperatures. The vent discharge temperatures were obtained from test data. However, no anemometer was put in the HVAC circuit to read out the exact airflow. Anemometer was subsequently put during the cold tunnel tests. Vent airflows were estimated from blower current and power readout with the mode valve set at vent full cold. From test data approximately 95-100 cfm airflow at 17 C was estimated to be entering the cabin from the HVAC vents when the automatic climate control system was set at 29C for ambient at 88F/55RH, 500 w/m^2 solar load set temperature condition in outside air. This amount of airflow is small because of warm cabin set point as opposed to 22C comfort set point when the blower runs at higher power. At 29C set point in steady state the blower operated at less than 5V. The convection velocities on the occupant for 95cfm airflow i.e. 24 cfm per each vent are much smaller than when the total airflow in the cabin is 200 cfm and above. At 24cfm from each vent discharged at 17C when the air reaches the occupant in 29C cabin is much warmer and air velocities around the occupant are small. The heat transfer coefficients are thereby small at 95cfm compared to 200-250cfm. Thus for the baseline case(spot cooling off) in energy efficiency mode the entire cabin operates at a weak to mid convection regime compared to strong convection regime observed during transients and lower in-car set point operation. From CFD modeling point of view much higher accuracy is needed to capture accurately all the convection coefficients, radiation exchange in the cabin and with the occupant, buoyancy effects, thermal mass effects, etc., etc. With spot cooling, the convection coefficients in the near field of the occupant even with low overall airflow are much higher.

A simplified sunroof CFD model was added to the simulation to better mimic the roof thermal conditions and comprehend the effect of hot roof on occupant head. It is important to capture the hot thermal conditions due to glass in the sunroof, since the occupant discomfort; especially in the head for warm in-car condition is significant. The sunroof model consisted of 8 mm thick low transmissivity glass exposed to convection heat transfer on the outside at 30 mph. Underneath the glass was a 10 mm thick air cavity exposed to natural convection heat transfer. Next to the air layer was 2 mm thick light colored beige cloth as found in the vehicle which was used as sun shield. The cloth was exposed to convection heat transfer of the cabin. The natural convection of the air cavity was not modeled; instead its effect was approximated by pure conduction through air at enhanced conductivity. Heat transfer from the roof to the cloth was simplified through conduction only, no radiation exchange between cloth and glass was modeled. For lack of appropriate information, the radiation load estimated by CFD from the roof ranged from 120w-250w, depending on solar load. One single roof temperature was measured in the middle of the sunroof with the thermocouple mounted in the center of the cloth on the cabin side.

Mostly, the two front passenger breath temperatures were used to correlate the CFD predictions with test data for the baseline case when spot cooling was turned off. When spot cooling was turned on, the flow field being so complex, that conventional definition of breath temperature and the usefulness of breath temperature metric to reflect cabin condition got minimized. With spot cooling the breathing zone temperature can be significantly different from the average in car temperature. The cooler chest cooling/face cooling air after entrainment reflected close to the breath temperature.

4.2. Results of Baseline CFD simulation

Fig 11 shows the velocity contour around the passengers 7.5mm in front of the passenger skin for quad combination spot cooling when conventional HVAC was turned off. As is evident from the velocity contours in Fig 11 the chest & face cooling nozzles were quite efficient in distributing cold air around the passengers. Airflow velocities both in their magnitude and distribution were quite uniform over the entire upper body part in cooling mode.

Fig12 shows the airflow path lines for the baseline case when spot cooling was off with 24 cfm flowing through each vent. In the CFD simulation as evidenced in Fig 12 the vents were directed towards the shoulder for the driver, and slightly skewed to one side for the passenger. The intent was to observe difference in breath temperature between the two. Fig 13 shows the velocity vectors in the mid-plane of the driver dummy. A number of complex re-circulatory cells are visible from the velocity vector plot. The effect of the box on the induced flow field is visible. Fig 14 shows how the high velocity air hits the upper body of the driver in the facial region contributing to cooler breath temperature. Even though the in-car condition is set for 29C, the breath level temperature is around 25-27C in the front of the vehicle for baseline case when no spot cooling was employed. Fig 14b shows the strong gradient in breath level temperatures in front of the occupant.

Fig 15 shows the pathlines when spot cooling was turned on in conjunction with conventional HVAC air. From this run it was observed that the front breath temperatures were lower

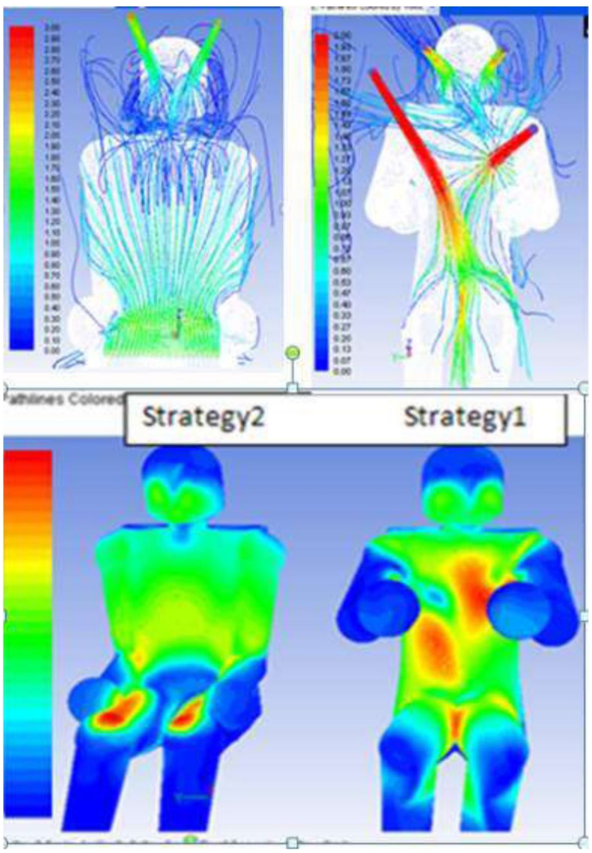

Fig 11. shows comparison of velocity field for combination spot cooling only (HVAC off)-Strategy1 (at 30% HVAC air) & Strategy 2(50% HVAC air)

by 1-3 C than the baseline case when spot cooling was off. As expected the air temperatures around the occupant were much closer to the nozzle air temperature. Fig 16 shows temperature & velocity contours for tri-combination around the occupants.

5.0. Results & Discussion

In Fig 17 is shown the comparison of spot cooling enhanced HVAC system with traditional HVAC operation. With conventional HVAC automatic climate control system operating in comfort set mode at 24C, in steady state from blower power data in the tunnel it was estimated that airflow was about 105 cfm. The average vent discharge temperatures were measured to be 5C. At this condition the cabin was maintained at around 24C EHT(equivalent homogeneous temperature), a metric commonly used to map non-homogeneous convective conditions to homogeneous, quiescent cabin condition.

For equivalent comfort that was achieved at 24C EHT set point described above, when spot cooling was enabled, the HVAC vents discharged only 95 cfm air at 17C to maintain cabin at 29C EHT. With spot cooling on the HVAC air had considerably higher discharge temperature. The air discharge temperatures with spot cooling for CFD model were set the same as tested in the tunnel. With spot cooling 60% higher velocities on the upper body part around the face, chest, pelvis, abdomen, back is attained. This results in higher heat transfer coefficient on targeted body part. Seat cooling was not simulated in this CFD analysis, though enabled in the

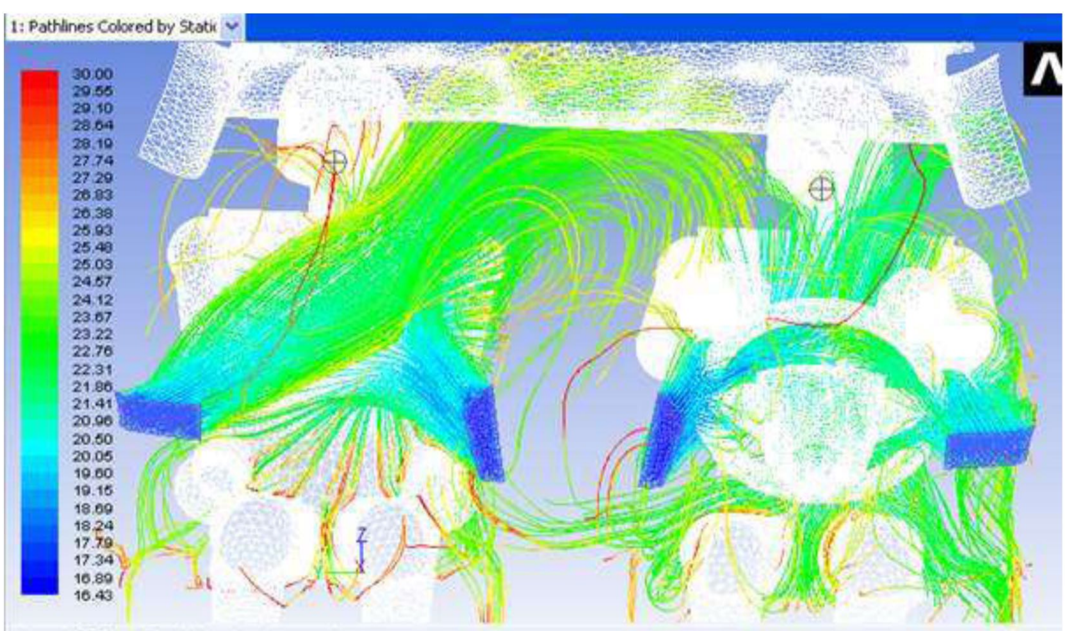

Fig 12. Airflow path lines originating from front HVAC AC vents.

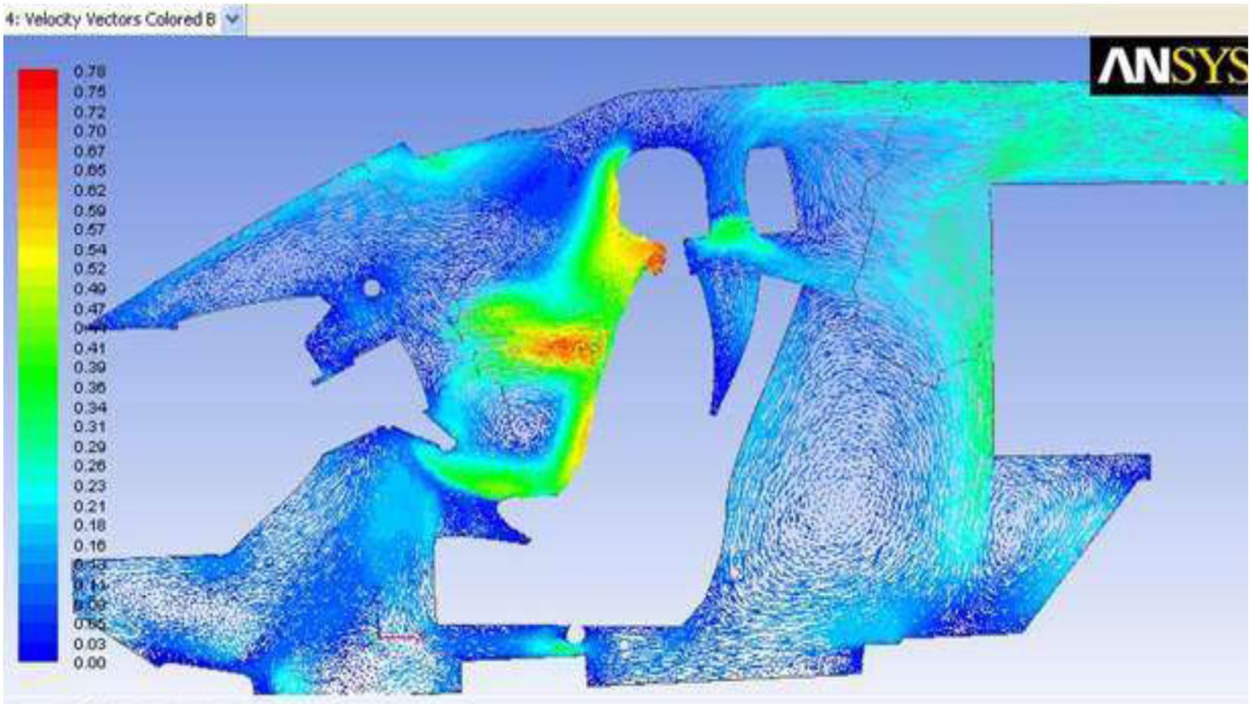

Fig 13. Air Velocity Vectors at mid-plane of the driver for Baseline case (Box model)

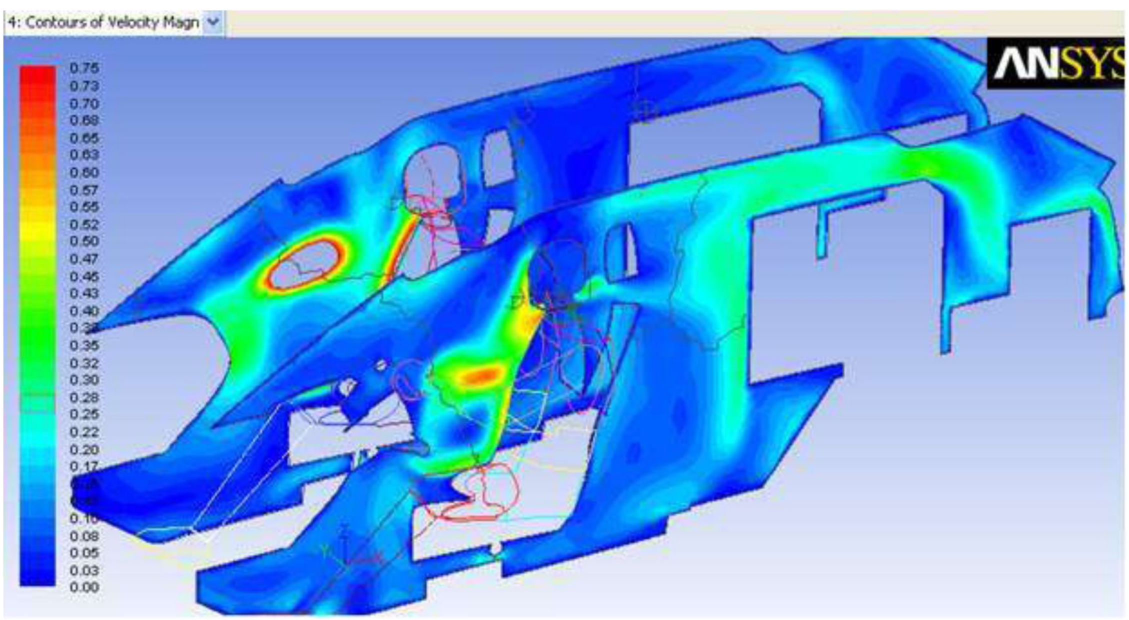

Fig 14. Velocity contours around occupant's face from front HVAC AC vent airflow for Baseline

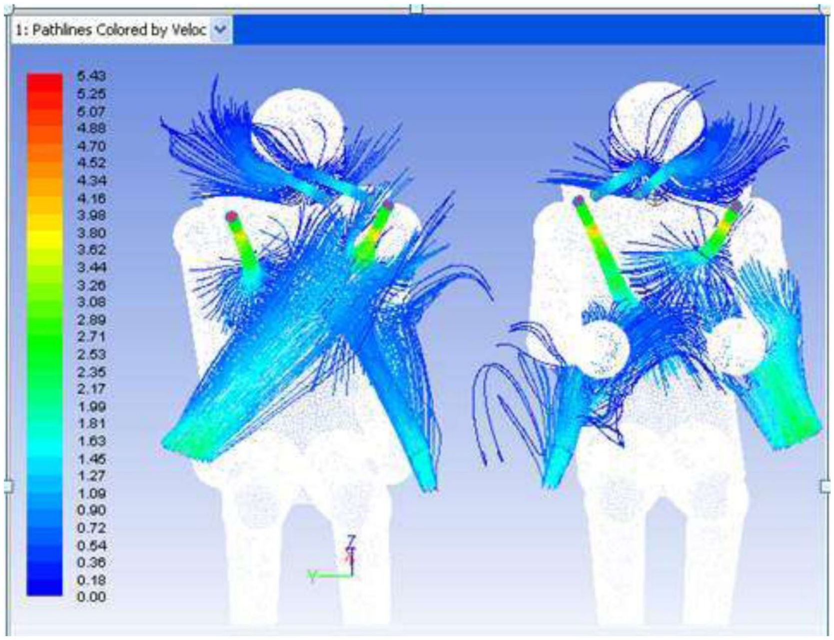

Fig. 15. Airflow pathlines for tri-combination (Face+Chest_Seat) cooling & HVAC air

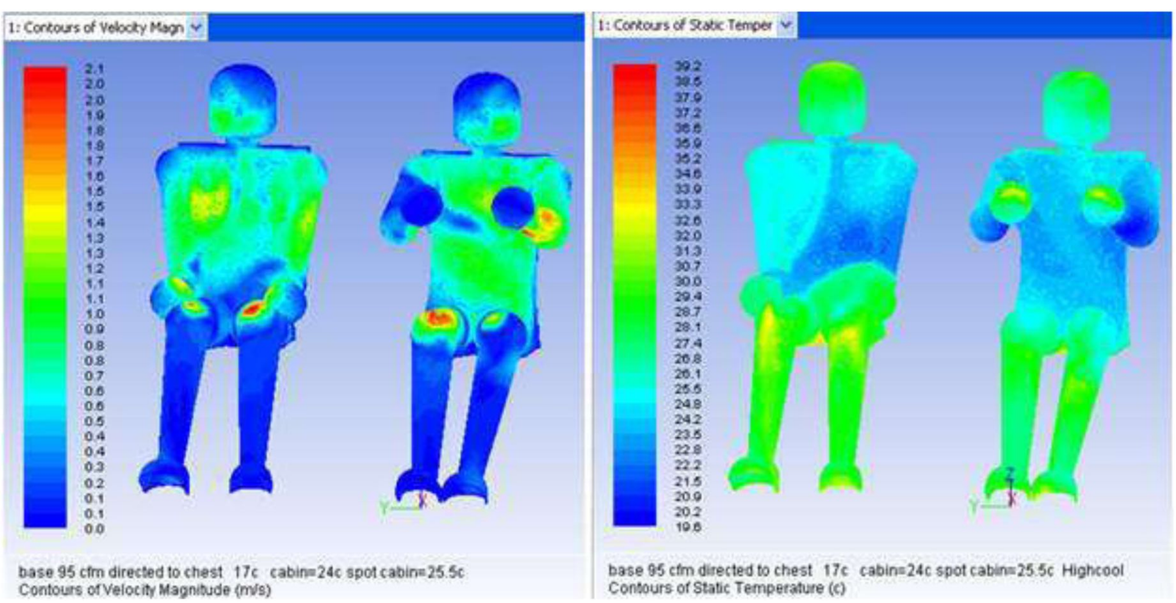

Fig 16. Temperature & Velocity contours for Tri-combination around the occupants

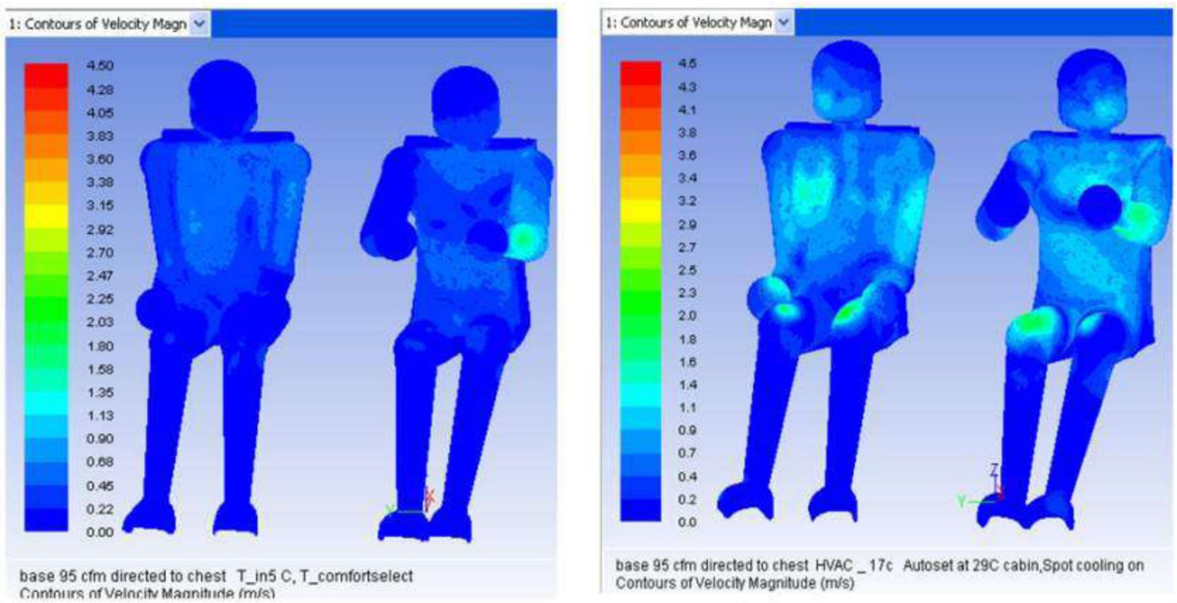

Fig 17. Comparison of Velocity contour for HVAC only (Automatic Climate control at Set point for comfort 24C) & elevated HVAC set pt=29C and quad-combination spot cooling on.

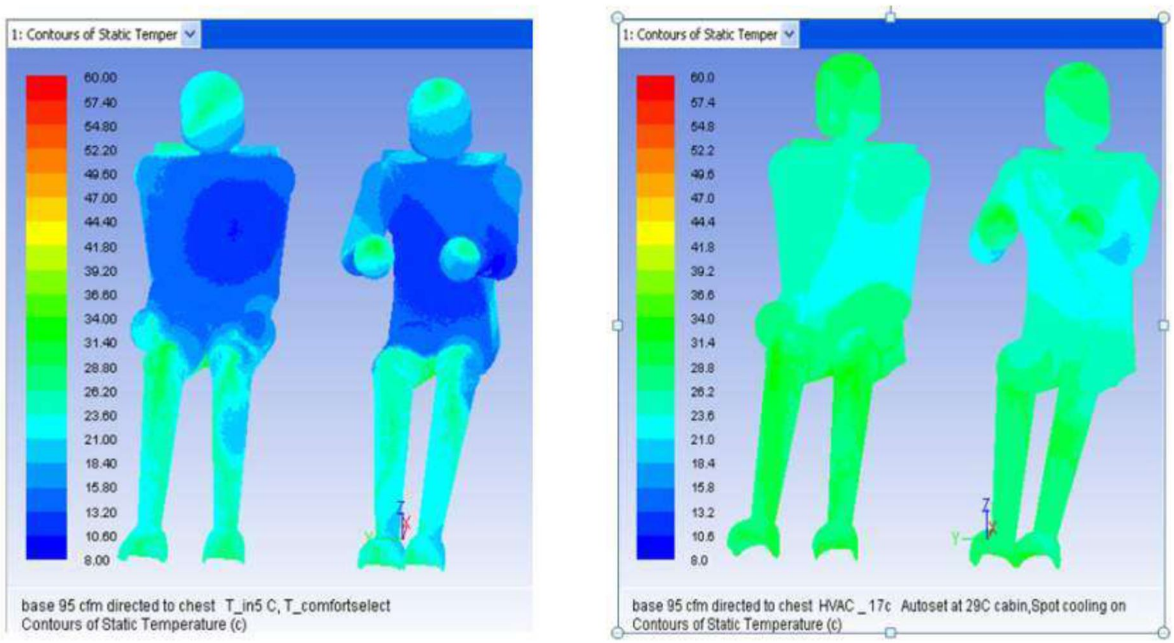

Fig 18. Comparison of Temperature contour: left- HVAC only (Automatic Climate control at Set point for comfort 24C) & right-elevated HVAC set pt=29C and quad combination spot cooling on.

tunnel tests. From airflow and heat transfer analysis it was estimated that seat cooling was performing almost 25% of the total heat transfer in steady state.

Fig 18 shows the variation in air temperature 7.5 mm in front of the occupant. Even though visually the air temperature for spot cooling individuals look warmer than the baseline case, the net effect were both same. For almost equivalent comfort the vent discharge temperature were 12C cooler in the comfort set point of 24C vs. energy saving mode of 29C. With spot cooling the air temperatures around the occupant were 22-24C. The occupant skin temperature is about 35C. In the baseline case with 5C discharge temperature of air around the occupant was around 14-15 C. Since total heat transfer is the product of heat transfer coefficient time's temperature difference, with spot cooling the higher htc than base line compensated for the drop in skin to air temperature differential. In fact, the occupant was more comfortable than the equivalent comfort baseline case. This is because chances of over cooling a body part were eliminated by discharging optimum temperature of air from the nozzles. This was validated by tunnel comfort rides. In the baseline case, as evidenced in Fig.18 the driver hands and lower arm

Table 1. Comparison of Glass Temperatures predicted by CFD with test data.

HVAC AC on 105cfm for all cases	**Contents**	**Glass Temperatures of SRX Tunnel Cabin at 88f/55RH ,500w/m^2 Solar Load**						
		ff Windshield C	ff Swd C	rr Swd C	rr Windshield C	Roof C		
CFD_Baseline_box	rr Seat off, Box in	45	36	36.5	42	36	Spot cooling off	
Test		47.7	33	32	42	42		
CFD_Baseline _full cab	rr Seat on, no Box	44	34	34.5	42.5	40	Spot cooling off	
CFD_tricombination	rr Seat off, Box in	44	35	36	42.5	34	Face+Chest+Seat on	

Table 2. Comparison of front Breath Temperatures predicted by CFD with test data

	T_driver front Breath		**T_passenger front Breath**		Coments	
HVAC on all the time	ff -Driver-T_breath1	ff -Driver-T_breath2	ff -Passenger-T_breath1	ff -Passenger-T_breath2		
Test	27.4 C	27.2 C	27.2 C	27.1 C		
CFD_Baseline_box cabi	25 C		27 C		Spot cooling off	
CFD_box_Spotcooling	24.5 C		25 C		Face & Chest Cooling on	

being exposed to direct draft of very cold air (5C), precipitated over cooling of the lower arm, a possible vasoconstriction situation if exposed for extended period in the same position. Comfort ride data confirmed that. However, human adaptive behavior moving the arm away from the flow path should remedy the situation.

6.0. Validation

The primary metric of significance that was used to compare the accuracy of the CFD prediction with test data was the front driver and passenger breath level temperatures. In test data, the thermocouples were hung a little higher and in front than the corresponding distance for 50 percentile individual. Table 1 and Table 2 show the comparison of all the glass temperatures and breath level temperatures with test data. Overall the correlation was found to be quite good. The breath level temperature variations with test data were within 2C. Quite a few CFD runs were conducted to evaluate the sensitivity of HVAC vent directivity towards the passenger on front breath temp. From the analysis it was found that at low HVAC flows the front breath temperatures can vary by as much as 5 C due to different airflow directions from AC vents towards the passengers. Vent directivity is an important set condition that needs to be considered. However, during the tunnel tests the HVAC vents were inadvertently kept flexible. Passengers could direct them for maximum comfort or leave as is during the comfort rides.

The AC vent airflow into the cabin were not measured during the tunnel tests, so the airflow at a particular in-car set point was estimated from indirect HVAC module pressure drop and fan power data. 95 cfm airflow was estimated for 88F/55RH under solar load test condition to obtain 29 C in-car conditions. No anemometer was placed in the HVAC air-inlet plenum. The AC vent outlet air temperatures were measured in the car within +−0.5C. Figure 19 shows the temperature of the cabin interior for the baseline case for 29C EHT condition. The IP and the front windshield were the hottest surfaces in the cabin, causing significant radiation exchange with the front occupants. The seat temperatures were cooler than test, as the seat model was simplified significantly to be adiabatic wall.

The correlation for all the windows- i.e. the glass temperatures with test data as shown in Fig 20 was quite good within 1.5C. The correlation for the very important roof temperatures were much better for the full cabin model as shown in Fig 21 than the CFD model with the TE simulator box with the rear seat removed discussed in Table 2. Without the sun roof model the roof interior was at much lower temperature. With sunroof model the roof temperature was higher by 6C from CFD prediction for box cabin. The roof temperature was within 2C compared to test for the full cabin with rear seat on. The side window was within 1-3C with test. Some of the variation could be attributed to vent directivity. Also the per CFD results there was quite a bit of temperature gradient in the windows. During tests only one point was measured on the glass.

In the box CFD model main variation in the roof temperatures predicted by the CFD box model was attributed to higher velocities around the roof than the full cabin model. CFD probably over-predicted the in-car airflow over the roof cover causing it to run cooler than the full cabin model. However, with a simple implementation of the Sunroof cfd model, the roof temperature got closer to test data than without the sunroof enhancement. The sunroof model helped to get a better estimate of the radiation load on the passenger. In the actual vehicle there must have been other flow paths going around the seats to the rear, than lot of it going over the roof in the box cabin model.

From overall the comparisons of cabin it was found that the cabin average temperatures were predicted to be around 2.5C cooler than tunnel measurements. A major source on this difference is due to modeling simplification used for spot cooling. Most of the early CFD work was done to do design

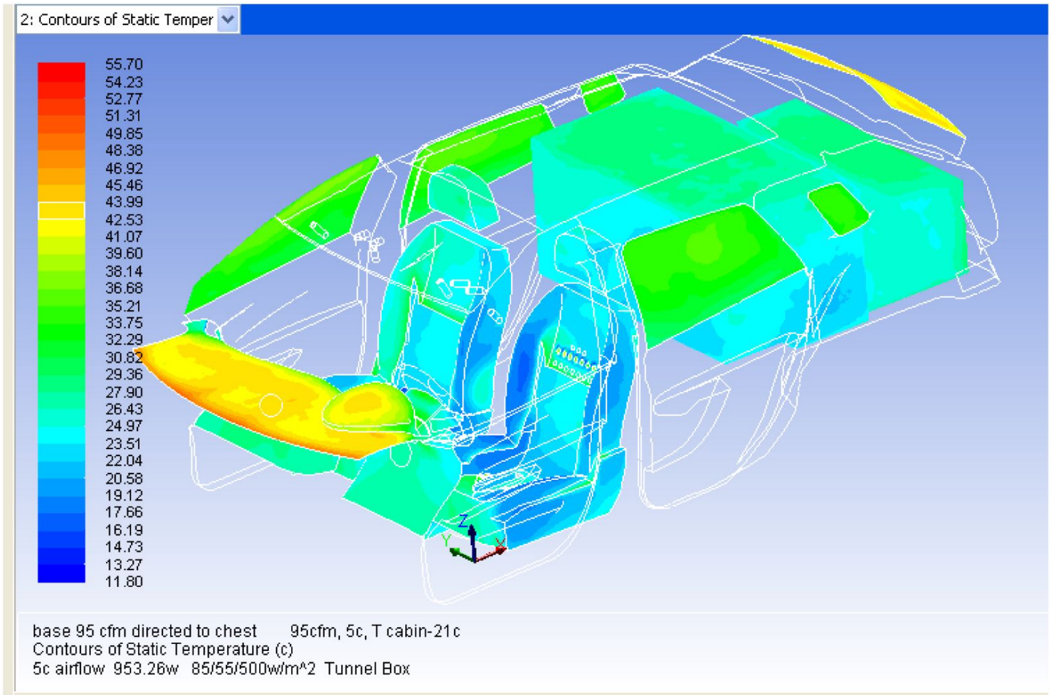

Fig 19. Temperature Contour for cabin walls Automatic Climate control at Set point 29C

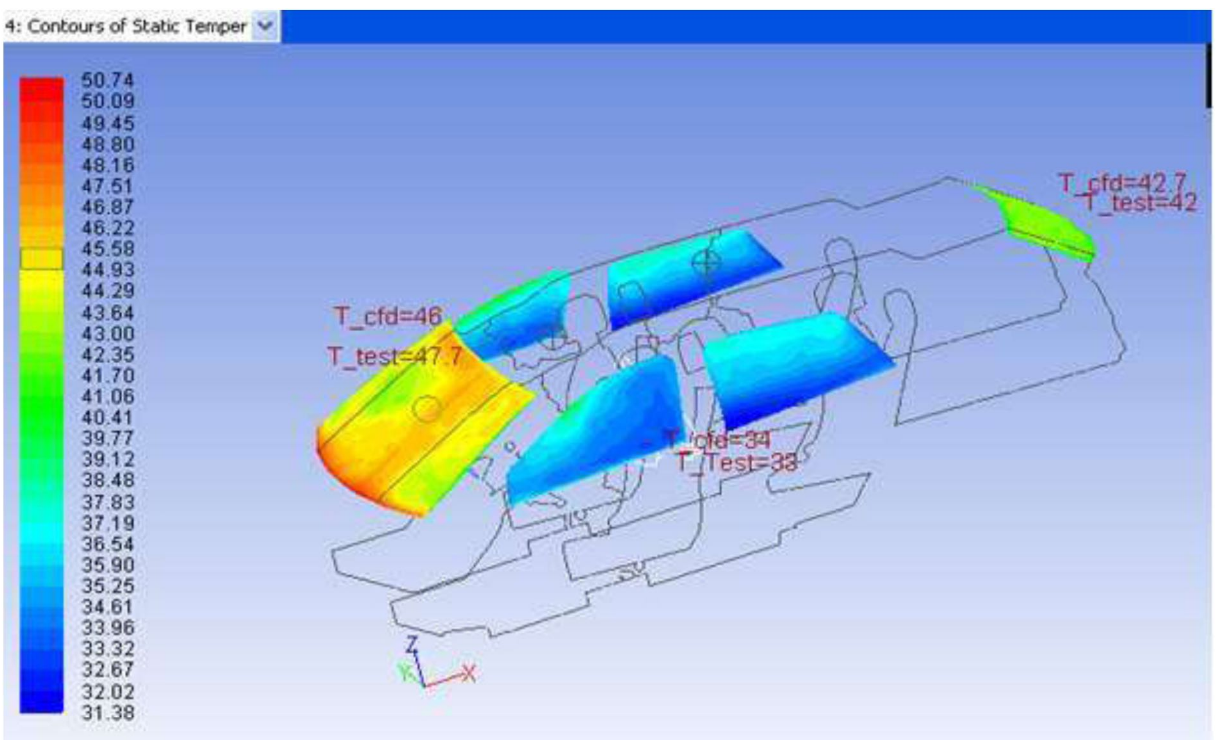

Fig. 20. Comparison of glass temperatures predicted by CFD with test data for full cabin.

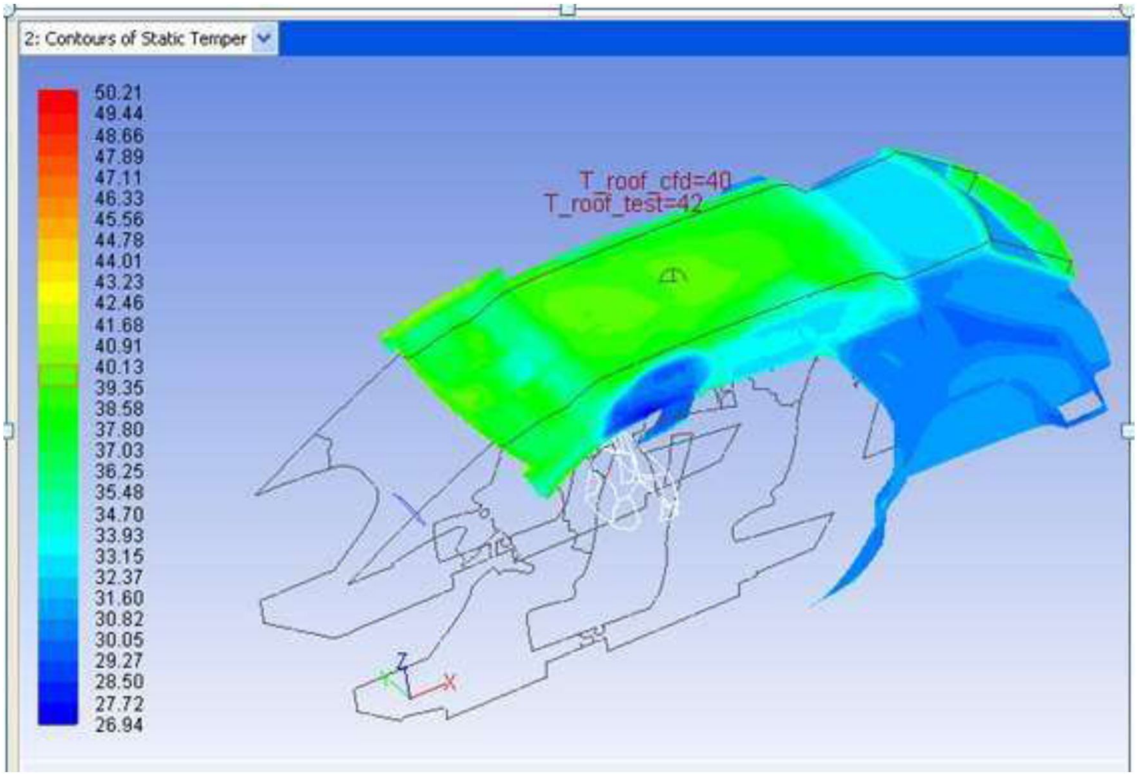

Fig. 21. Comparison of Roof temperatures predicted by CFD with test data for full cabin

the spot cooling system and not perform virtual tunnel tests. For this purpose doing the simulation with spot cooling air in outside air mode was quite adequate as HVAC was mostly turned off or total HVAC airflow was low. However, in the actual implementation of spot cooling in the vehicle and during tunnel tests, spot cooling air was used in re-circulatory mode. In outside air mode spot cooling air was always vented out in steady state which caused a cooler cabin in CFD. This correlation could be improved if spot cooling airflow is modeled in re-circulatory mode. Modeling spot cooling in recirculation mode is doable, but adds meshing and solution complexity and longer run time. More accurate seat modeling that will accurately capture the seat thermal mass effects and make the seat run warmer which should also improve the correlation. Scope exits to improve the body heat source into the model. All these improvements can be part of future work. To capture transient effects the model needs to be refined lot further; more definition is needed around the material properties. This could be attempted once good transient tunnel data is obtained mapping the in-car surfaces and cabin conditions.

There can be many factors contributing to variation between test data and CFD prediction. Uncertainty in HVAC airflow, directivity of the HVAC vents, uncertainty in material properties, radiation intensity and radiation spectra variance of the tunnel lights from the actual regular and diffuse solar radiation outside the car are thought to be the primary variables. Engine heat leak and exhaust system heat leak into the cabin need to be accounted for the total heat budget. For design purposes of complex real life in-situ automotive HVAC systems with spot cooling the accuracy of CFD prediction is quite good. It is able to rank order designs quite effectively.

7.0. Summary and Conclusions

CFD analysis was very effectively utilized to design the spot cooling system. In the energy saving mode since the in-car set point is elevated, the total HVAC airflow and the enthalpy delivery into the cabin is smaller compared to all the other heat loads like solar load, convection heat loss, etc. Since the cabin operates in weak convection regime at 29c set point compared to 22c set point, the modeling accuracy required for the CFD model is high. The simulation technique was able to capture the relevant physics quite adequately. Of all the individual spot cooling strategies, face, chest and lap cooling were found to be most efficient in cooling mode. The quad-combination of face, chest, lap, and seat cooling was found to be most robust to all kinds of cabin and occupant population variation. However, chest, face and seat cooling were quite efficient too. With spot cooling it was analytically found that under ideal conditions of nozzle locations and airflow delivery, it is possible to achieve similar cooling with almost about thirty-to fifty percent of conventional HVAC airflow per person. Validation was done post-factum after the tunnel to correlate the CFD model with tunnel tests. Overall agreement of CFD predictions in steady state was found to be very good. Much more work is needed for transient cabin simulation both without and with spot cooling at warm in-car conditions.

References

1. Arens, E., Zhang, H., Huizenga, C., 2006, Partial and whole body thermal sensation and comfort-Part1: Uniform environmental conditions, J. Thermal Biology 31, 53-59.
2. Arens, E., Zhang, H., Huizenga, C., 2006, Partial and whole body thermal sensation and comfort-Part 2: Non-uniform environmental conditions, J. Thermal Biology 31, 60-66
3. Kaushik, S., Chen, K.H., Han, T., and Khalighi, B., Micro-Cooling/Heating Strategy for Energy Efficient HVAC System, 2011-01-0644
4. Chen, K.H., Kaushik, S., Han, T., Ghosh, D., Wang, M., Thermal Comfort Prediction and Validation in a Realistic Vehicle Thermal Environment, 2012-01-0645

Acknowledgement

This material presented in this paper is based upon the work supported by the Department of Energy (National Nuclear Security Administration) under Award Number DE-EE0000014. The authors are thankful to DOE for the opportunity to execute this project. The authors wish to thank Garry Warren for all the tedious and detail designs for CFD and building parts. The authors are thankful to Mark Zima, Tim Craig, and Prasad Kadle for all cooperation, discussion and all the encouragement in executing such a huge task in time. A special thanks to the Delphi tunnel team.

Contact Information

Debashis Ghosh
Delphi Automotive, Lockport, NY-14094
Phone no.: 716-438-4923;
debashis.ghosh@delphi.com

Disclaimer