"Performance optimization of an air- conditioning system of automobile vehicle by changing air outlet location."

Aditya Singh Lodhi *, Dr. G.S Sharma **, Mukesh Dole***

* Research scholar, Department of Mechanical Engineering, Lakshmi Narain college of technology and science (RIT) , Indore.

**Principal, Department of Mechanical Engineering, Lakshmi Narain college of technology and science (RIT), Indore.

***HOD, Department of Mechanical Engineering, Lakshmi Narain college of technology and science (RIT), Indore.

Abstract: Air vent location in an automobile has a significant impact on thermal comfort of cabin. In the current research the location of air vents has been changed from all 4 in front dashboard (Design 1) to 2 fronts and 2 rears above passenger seat (Design 2). Transient thermal analysis is performed using ANSYS CFX 18.1. With the assistance of CFD it is demonstrated that rear cooling air vents give a superior air comfort to passenger. Air flow design has been considered and contrasted and the consistent air vent at different location which covers the maximum area under the passenger car cabin. To direct air flow inside the car cabin air vent at front side on dashboard and rear side of the cooling vane development has been reproduced. Due to change of location of air vent air flow and temperature control has improved towards the back passenger. Cabin chill off examination with the rear cooling side vents has been done to study the impact of cooling inside the Cabin. A slightly faster cool down in seen with the design 2 air vents at different locations compared to design 2 all air vent located at the dashboard only. The initial cabin temperature for analysis is considered to be 313K.

Key Words: Airflow characteristics, CFD, HVAC, EPA, TDV

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 considered 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. Be that as it may, the comfort level being subjective it is difficult to set conclusive levels. The main controlling measures are airflow velocity, cabin temperature and relative moistness. 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 extent of the ventilating framework is identified with the pinnacle heat load in the vehicle. Human thermal comfort is defined by The American Society of Heating Refrigeration and Air Conditioning Engineers (ASHRAE), as the state of mind that expresses satisfaction with the surrounding environment (ASHRAE Standard 55). The principal factors affecting human thermal comfort depends upon four physical environmental variables; the air temperature, its relative humidity, the mean radiant temperature, and the relative air velocity; in addition to two independent but related parameters, which are the active level provided by metabolism and the thermal insulation value provided the clothing.

2. LITERATURE REVIEW

Smyth et. al.[1] describes direct expansion (DX) refrigeration technology is almost exclusively used in Multitemperature transport refrigeration systems. Multitemperature systems use up to three evaporators, requiring large refrigerant charges and system pressure control to operate over a wide range of set-point conditions. Despite incremental design improvements over the past decade, environmental and control issues continue to arise with DX systems. Deployment of indirect refrigeration systems (IDX) offers an alternative approach to address these issues. Indirect systems can however suffer from performance penalties, where reduced cooling capacity and COP occur under certain operating conditions. One strategy, aimed at offsetting the disadvantage of reduced refrigeration capacity, is to incorporate an economizer circuit into the primary cycle of the IDX system. Economizer cycles can enhance the refrigeration effect of the primary refrigerant in the primary to secondary heat exchanger of the indirect system.

Smyth et. al.[2] describes an approach for control of an economiser cycle based on the use of economiser pressure as the primary control parameter. In the study, the economiser cycle was used to optimise a multitemperature indirect (IDX) transport refrigeration system, where hydronic secondary loops were utilised. In transport refrigeration applications, IDX systems can offer the potential to address a number of important environmental and control issues associated with direct expansion (DX) systems. IDX systems may also give rise to reduced capacity and COP through increased compressor pressure ratios associated with the hydronic secondary circuit and power requirements of the liquid secondary pumps. One approach by which this issue can be addressed is through use of an economiser cycle, which provides a mechanism for performance enhancement by augmenting the refrigeration effect of the primary refrigerant, in the primary to secondary heat exchanger of these systems. Previous work ascertained that by control of the mass-flow injection ratio, an economiser cycle can be used to optimise indirect multi-temperature systems for a wide range of diverse operating conditions.

Finn et. al.[3] describes a mathematical model of the defrost process for a finned-tube air chiller, utilised as a heat exchanger in a secondary loop multi-temperature transport refrigeration system, where an antifreeze mixture is deployed as a sensible secondary working fluid. Two defrost modes are modeled: an electric mode which effects defrost by localised resistance heating of the chiller secondary working fluid, and a hot gas primary circuit mode that indirectly heats the secondary working fluid by means of a primary to secondary heat exchanger. The model, which was implemented using the Engineering Equation Solver (EES), is based on a finite difference approach to analyse the heat transfer from the secondary working fluid, through a single finned heat exchanger section, to the frost. An iterative scheme is used to integrate for the overall heat exchanger, considering temperature glide associated with the secondary working fluid. The overall heat exchanger model is incorporated within a system defrost model, which allows the entire defrost process to be modelled.

Winkle et. al.[4] has been growing in recent years due to the high direct global warming potential of common HFC refrigerants. Despite the environment-friendly characteristics of CO2 as a refrigerant, due to high heat rejection temperatures and trans critical operation, CO2 cannot match the high energy efficiency associated with current HFC technology. Thus, additional measures must be taken to achieve high COP when using CO2. One approach is to use CO2 as one of the fluids in a cascade system along with a HFC refrigerant as the high side fluid. Such systems may have roughly 75% less HFC refrigerant charge, and the global warming potential is reduced compared to a baseline system using only HFC refrigerant. When used as a second fluid in a cascade system, the CO2 cycle remains in the subcritical region, thus increasing the cycle's COP. In this paper an approach to model cascade systems is presented. The model is validated using experimental data for a R404A/CO2 cascade system and results are discussed.

Yamasaki et. al.[5] describes the transcritical refrigeration cycle utilizing CO2 as working fluid which is composed with Gas cooler, Intercooler, Suction Line Heat Exchanger, Capillary tube and Rolling Piston type 2-Stage CO2 Compressor. The adoption of the Inter cooler between 1st discharge and 2nd suction reduced the 2nd discharge gas temperature. The adoption of the Suction Line Heat Exchanger enabled to have a sufficient superheat of 1st suction. The adoption of capillary tube as an expansion device helped the system simplicity.

Pfafferottet. al.[6] presents the current results of the development of a Modelica library for CO2-Refrigeration systems based on the free Modelica library ThermoFluid. The development of the library is carried out in a research project of EADS Airbus and the TUHH and is focused on the aim of getting a library for detailed numerical investigations of refrigeration systems with the rediscovered refrigerant carbon dioxide (CO2). A survey of the concept of an integrated cooling system on-board of airliners, the used modelling language Modelica™ and the developed CO2-Library is given and the modelling of CO2Heat exchangers is described. A comparison with steady state results of heat exchangers is presented showing a very good agreement. The presented transient simulation results show the expected trends, but the models have not yet been validated with transient experimental data.

3. OBJECTIVE

Rear Ac vents helps the car cool quiclkly. Most of the car without Rear Ac vents comes the passengers in front by the time is fully cools the car the destination may have reached. So the cars with rear Ac vents gives the passengers in back more comfort and the temp adjustments gives them more comfort by adjusting the level of cooling of their own without making it hard for the one sitting in the front. It helps the rear passenger of the car to feel comfort. Entire Car's temperature gets changed rapidly due to this additional feature. Negative side, the price of the vehicle rises up.

4. METHODOLOGY

The research analysis is performed using Computational Fluid Dynamics technique where CAD modeled is developed and imported in ANSYS CFX solver where suitable boundary conditions is applied and solved. The analysis involves three stages, first is pre-processing, solution and post processing stages.

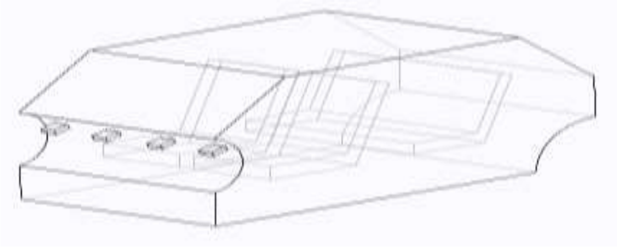

Figure 1: CAD model of car with 4 front vents

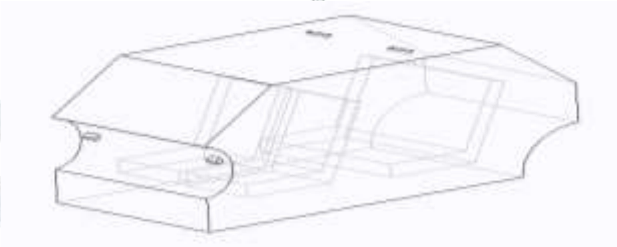

Figure 2: CAD model of car with 2 front and 2 rear vents

A pre-processor is used to define the geometry for the computational domain of interest and generate the mesh of control volumes (for calculations). Generally, the finer the mesh in the areas of large changes the more accurate the solution. Fineness of the grid also determines the computer hardware and calculation time needed. The solver makes the calculations using a numerical solution technique, which can use finite difference, finite element, or spectral methods.

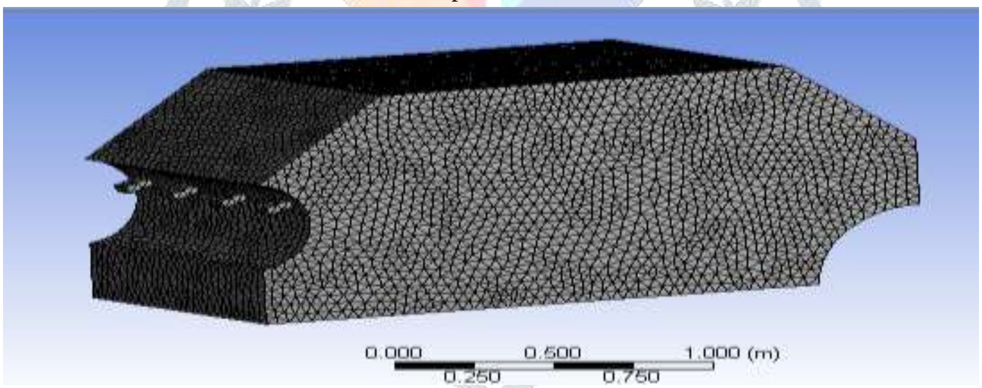

Figure 3:Meshed model of car with 2 front and 2 rear vents

Most CFD codes use finite volumes, which is a special finite difference method. First the fluid flow equations are integrated over the control volumes (resulting in the exact conservation of relevant properties for each finite volume), then these integral equations are discretized (producing algebraic equations through converting of the integral fluid flow equations), and finally an iterative method is used to solve the algebraic equations. The post-processor provides for visualization of the results and includes the capability to display the geometry/mesh, create vector, contour, and 2D and 3D surface plots. Particles can be tracked throughout a simulation, and the model can be manipulated (i.e. changed by scaling, rotating, etc.), and all in full colored animated graphics.

5. ANALYTICAL CALCULATIONS

In the present study, a model experiment is performed in order to clarify the airflow characteristics of a car cabin. In addition, this study provides high precision data for a benchmark test using the CFD (Computational Fluid Dynamics) analysis method. Initially, the study focuses on the ventilation mode that describes the flow field in the car cabin obtained from compression data obtain from CFD by changing the location of air vents.

- Mass flow rate: As the blower is not considered for simulation, the mass flow rate is given to HVAC inlet. the air velocity is between 0.1m/s~0.4 m/s..
- Time step size (Δt): The time step size is the time in seconds for which the simulation progresses. It is calculated from the smallest cell size and initiated velocity and is given as follows, the calculated time step size is 0.03 second.
- Number of time steps: The number of time steps for which the flow is to be calculated till the solution reaches convergence. In transient dynamic meshing models time step is of crucial importance and has to be calculated from the moving zones.

Heat Load calculation Procedure

The net heat gain by the cabin can be classified under different categories. The total load as well as each of these loads may depend on various driving parameters. In the following, each of these load categories are presented and discussed. Some of the correlations used here for general validation of load calculation. The summation of all the load types will be the instantaneous cabin overall heat load gain. The mathematical formulation of the model can thus be summarized as

$$Q_{Total} = Q_{Metabolic} + Q_{Solar} + Q_{Equipment} + Q_{Fresh\ air}$$

Metabolic Load

The Metabolic activities inside human body constantly create heat and humidity (i.e. perspiration). This heat passes through the body tissues and is finally released to the cabin air. This amount is considered as a heat gain by the cabin air and is called the metabolic load. The metabolic load can be calculated by

$$Q_{Metabolic} = M \times A$$

Where,
M- Metabolic Load, W/m2
A- Area of the body, m2
Which can be calculated by the Dubois Surface AreaA_{Du};

$$A_{Du} = 0.202 \times W^{0.425} \times H^{0.725}$$

Where,
W- Weight of the person, kg
H- Height of the person, m
For the calculation of the Metabolic load consider five persons in the car at a time and each person weight 90 kg and height 6 ft then area of each person same which is

$$A_{Du} = 2.052m^2$$

Solar load

The heat gain due to solar radiation is a significant part of the cooling loads encountered in vehicles. The radiation which comes is normal solar radiation and it is divided into direct, diffuse and reflected radiation loads.
Normal Solar Radiation

$$I_n = 1082 \times e^{\frac{0.182}{sin\beta}}$$

Where, $\beta = altitude angle$

$$Sin\beta = cosl \times cosh \times cosd + sinl \times sind$$

l = latitude

h = hour angle
d = sun declination

$$d = 23.47 \times sin^{\frac{360(284+N)}{365}}$$

N is the day of the year counted from 1 January

$$I_n = 895.20\ W/m^2$$

Direct Radiation

$$I_D = I_n cos\theta$$

$I_n = normal solar radiation$
θ = angle of incidence

$$cos\theta = cos\beta \times cos\alpha$$

α = surface solar azimuth angle

$$\alpha = 45 - h$$

$$I_d = 216.16\ W/m^2$$

Diffuse Radiation

$$I_D = CI_n F_{ss}$$

Where,

C Dimensionless coefficient for sky radiation of each month.

F_{ss} Factor between the sky and the surface.

$$F_{ss} = 0.5$$

$C = 0.121$ for may month

$$I_d = 54.159\ W/m^2$$

Total heat gain of the space

$$Q_{Solar} = Q_1 + Q_2 + Q_3$$

Where,

Q_1 Total transmitted radiation.

Q_2 Total absorbed radiation.

Q_3 Heat transmission by conduction, convection and radiation.

Total Transmitted Radiation

$$Q_1 = A(\tau_D l_D + \tau_D l_D)$$

Where,

A Expose area, m^2

τ_D Transmissivity of the direct radiation

τ_D Transmissivity of the diffuse radiation.

l_D Direct radiation, W/m^2

l_D Diffuse radiation, W/m^2

Total absorbed radiation

$$Q_2 = \frac{A \times \alpha \times I}{1 + \frac{f_o}{f_i}}$$

Where,

A Area of window glass and roof, m^2

α Absorptivity of glass and roof,

I total radiation, W/m^2

f_oOutside heat transfer coefficient, W/m^2 k

f_i Inside heat transfer coefficient, W/m^2 k

Heat transmission by conduction, convection and radiation

$$Q_3 = UA(t_o - t_i)$$

$$\frac{1}{U} = \frac{1}{f_i} + \frac{\delta}{k} + \frac{1}{f_o}$$

U Overall heat transfer, W/m2 k

A Area of glass, door, roof, m^2

δ Thickness of the material, m

K thermal conductivity, W/m k

$$Q_{total,solar} = 1235.96\ W$$

Equipment load - In a car only equipment is the music system.

$$Q_{equipment} = 80\ W$$

Fresh air load

Fresh air flow

$$V = R_p \times P_z + R_a \times A_z$$

R_p Outdoor airflow rate required per person

P_z Zone population

R_a Outdoor airflow rate required per unit area

A_z Zone area

$$V = 0.708\ m^3/min$$

Mass of fresh air

$$m_{f_a} = 0.0129\ Kg/min$$

Sensible load

$$SH = m_{f_a} C_{pm}(t_o - t_i)$$

$$SH = 0.1977KW$$

Total Load

$$Total\ Load = m_{f_a} C_{pm}(h_o - h_i)$$

$$TL = 0.4347KW$$

Latent Load

$$Latent\ Load = Total\ Load - Sensible\ Load$$

$$LL = 0.2370KW$$

Table 4.1: Total Load Summary

Sr. no.	Source	Sensible Load, W	Latent Load, W
1	Occupant	605.34	287.28
2	Solar	1235.96	--
3	Equipment	80	_
4	Fresh Air	197.7	237
5	Total	2119	524.28
Total Load		**2643.28 W**	

Cooling load = 2.63/3.5

Cooling Load = 0.75TR

Mass flow rate of refrigerant

$$m_a = \frac{coolingload}{h_{fg}}$$

At 6˚C of coil ADP

$$h_{fg} = 364.49\ kj/kg$$

$$m_a = 0.0072\ Kg/sec$$

Volume flow rate of refrigerant

$$V = m_a v_f = 0.00002451 m^3/Sec$$

6. RESULTS AND DISCUSSION

For steady state analysis the air vent velocity blowing at 17^0C in Maruti 800 cabin. The temperature distribution, pressure distribution plot is derived from ANSYS CFX. The temperature distribution shows better cooling characteristics and lower cabin temperature achieved along with much uniform temperature distribution. Rear passengers receive better cooling in design 2 as compared to design 1.

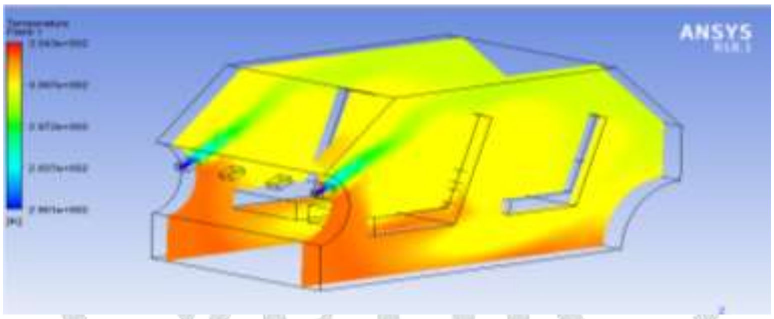

Figure 4: Steady state temperature distribution for design 1

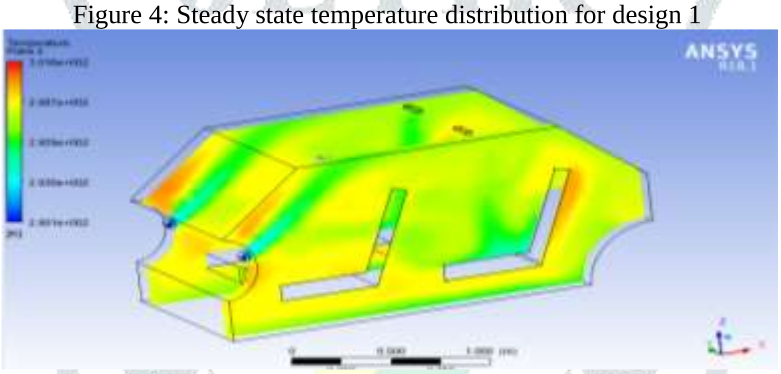

Figure 1.5: Steady state temperature distribution for design 2

Table 4.2: Average cabin temperature comparison

DESIGN TYPE	AVERAGE CABIN TEMP(K)
DESIGN 1	300.62
DESIGN 2	295.98

For transient state CFD analysis initial cabin temperature was considered to be 40^0C which 313K. The transient thermal simulation is performed keeping initial temperature at 313K and time of analysis was 50 counter seconds which corresponds to 10 minutes in real time situation.

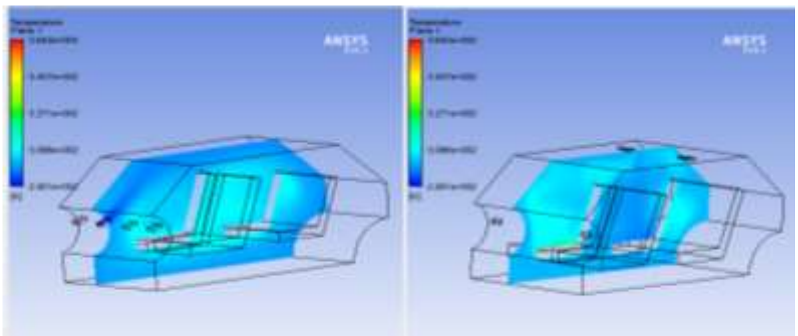

Figure 5: Temperature plots for design1 and design2 at 40Km/hr vehicle speed

After 50 counter seconds of cooling the temperature drop in rear passenger domain volume is significantly lower than front passenger domain volume as can be seen from figure 5 above. The average temperature of front passenger domain is in range of 298K to 307K while for rear passengers the average temperature of domain volume is 292K to 300K, which shows significant improvement in cooling for rear seat passengers thus improving thermal comfort.

Table 4.3 Car at Speed of 40km/hr Average Temp. Comparison

TIME	AVG CABIN TEMP DESIGN 1	AVG CABIN TEMP DESIGN 2
10	310.84	310.01
20	305.35	304.33
30	301.91	301.42
40	300.21	299.66
50	298.33	299.79

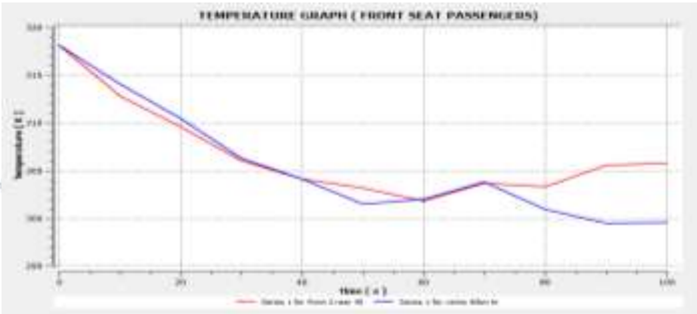

Figure 6: Temp vs time for front seat passengers, red for design 2, blue for design 1

Temperature versus time curve for front passengers both design 1 and design 2 shows temperature dropping at almost same rate upto 70 counter seconds and subsequently further drop is noticed in design 1 with 4 front vents whereas for design 2 temperature drop reduces and equilibrium temperature achieved in 100 counter seconds is higher for design 2 as compared to design 1

Figure 7: Temp vs time for rear seat passengers, red for design 2, blue for design 1

For rear seat passengers the temperature drop is higher for design 2 for almost all counter intervals as compared to design 1 evident from figure 5.12 above.

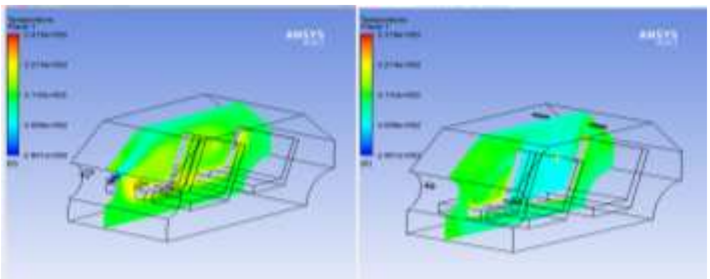

Figure 8: Temperature plots for design1 and design2 at 50Km/hr vehicle speed

The temperature drop achieved in 50Km/hr speed within 10 counter seconds of air flow inside cabin the temperature difference is noticeable between 2 designs. Rear passengers receive significantly higher cooling in design 2 with temperature averaging to

300K as compared to design 1 with temperature averaging 305K. In 20 counter seconds the cooling for rear passengers becomes almost stagnant showing similar temperature magnitude as in 10 counter seconds.

Table 4.4Car at Speed of 50km/hr Average Temp. Comparison

TIME	AVG CABIN TEMP DESIGN 1	AVG CABIN TEMP DESIGN 2
10	309.84	307.64
20	304.17	300.93
30	300.38	298.065
40	298.74	297.04
50	297.79	296.81

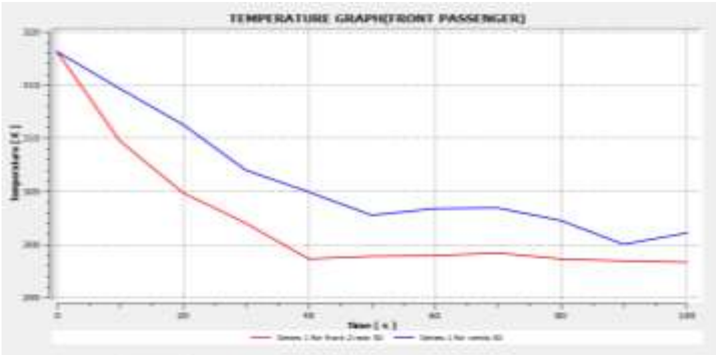

Figure 9: Temp vs time for front seat passengers, red for design 2, blue for design 1

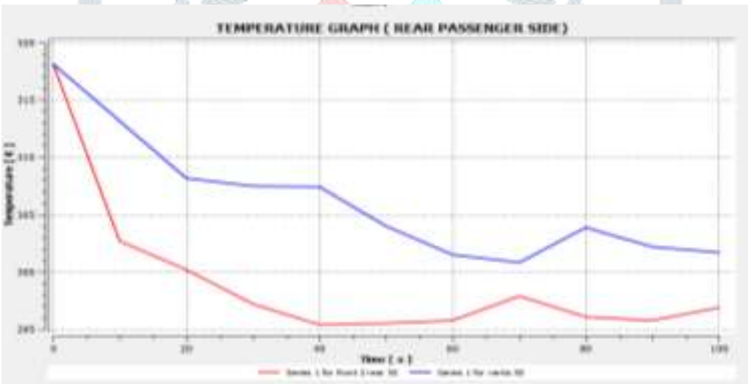

Figure 10: Temp vs time for rear seat passengers, red for design 2, blue for design 1

Temperature versus time curve for front passengers as shown in figure 10 for both design 1 and design 2 shows temperature drop is consistently lower in design 2 as compared to design 1 for all counter seconds. The temperature drop in design 2 has reached to 298K thereby maintaining constant level throughout. Similar trend can be seen in temperature drop curve for rear passengers where temperature drop is consistently lower in design 2 and reaching to an almost equilibrium point of 297K.

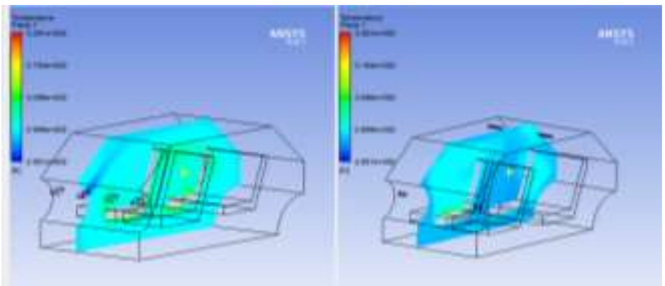

Figure 11: Temperature plots for design1 and design2 at 60Km/hr vehicle speed

The temperature drops further in rear passenger side in 30 counter seconds to averaging of 297K and for front seat passengers averaging 302.5K. In 40 counter seconds temperature further drops rear passenger domain averaging 296.8K and front passenger temperature of 301K. This shows design 2 is better in providing more cooling thereby providing more thermal comfort. Thereafter the cooling for rear passengers becomes almost stagnant showing similar temperature magnitude as in 10 counter seconds.

Table 4.5 Car at Speed of 60km/hr Average Temp. Comparison

TIME	AVG CABIN TEMP DESIGN 1	AVG CABIN TEMP DESIGN 2
10	308.56	305.93
20	302.79	299.18
30	299.14	296.74
40	297.94	296.10
50	297.28	295.58

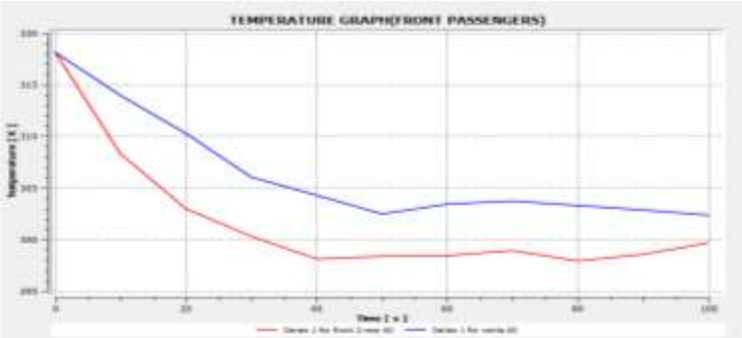

Figure 12: Temp vs time for front seat passengers, red for design 2, blue for design 1

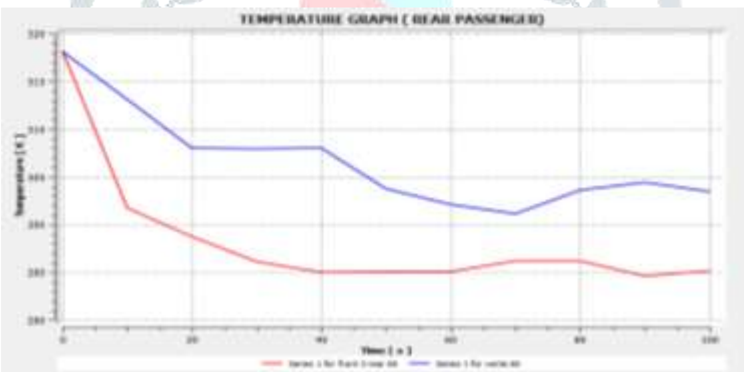

Figure 13: Temp vs time for rear seat passengers, red for design 2, blue for design

Temperature versus time curve for front passengers as shown in figure 5.26 for both design 1 and design 2 shows temperature drop is consistently lower in design 2 as compared to design 1 for all counter seconds. The temperature drop in design 2 has reached to 296K thereby maintaining constant level throughout. Similar trend can be seen in temperature drop curve for rear passengers where temperature drop is consistently lower in design 2 and reaching to an almost equilibrium point of 295K.

7. CONCLUSION

The cabin cool down test was carried out to check the cooling effect of air-conditioning with different vent locations. The test was carried for 50 counter second in CFD, which is equivalent to 10minutes. The cabin temperature from the start of the cool down cycle is monitored at the respective of air flow distribution. Temperature plot as a function of flow time for the air ventdesign 1 simulation has been compared with the design 2 data.

1. The cabin temperature builds up to 308 K during summer, and starts dropping down to a minimum of 297.28 K during the cool down cycle of design 1 all the vent are on dashboard.
2. The cabin temperature builds up to 308 K during summer, and starts dropping down to a minimum of 295.58 K during the cool down cycle of design 2 two vent are located on dashboard and others two on the top of rear seats.

3. A slightly faster cool down in seen with the design 2, air vents at different locations compared to design 2 all air vent located at the dashboard only. Cabin temperature settles down to approximate 295.58 K for design2 (2 air vent located at front and 2 at the rear of car) 297.28 K for design 1 (all vent located at front).
4. Under steady state conditions design 2 performed better with 295.98K average cabin temperature as compared to design 1 with average cabin temperature of 300.62K.
5. For front seated passengers design 1 offers more cooling as compared to design 2 due to more air flow due to location of all 4 air vents on front dashboard.
6. The CFD analysis is performed for different knob position 1 and 2. The knob position 2 delivers better cooling with higher outlet speed of 5.88 m/sec as compared to knob 1 position with 2.67m/sec air outlet velocity. This is applicable in both design 1 and design 2.
7. Vehicle speed has direct affect on cooling of cabin. CFD analysis is performed using 3 different vehicle speeds i.e 40Km/hr ,50Km/hr and 60Km/hr. Higher the vehicle speed, higher is the air flow over AC condenser which increases heat transfer rate and hence higher cooling is achieved.
8. CFD simulation is carried out to determine time to subside. The analysis is carried for 80 counter seconds and in design 2, cabin temperature subsides at faster rate as compared to design 1.

REFERENCES

[1]. Shane Smyth, Donal P. Finn, Barry Brophy, (2010), Performance Evaluation of an Economised Indirect Multi-Temperature Transport Refrigeration System, International Refrigeration and Air ConditioningConference,Paper 1095.

[2]. Smyth, Shane; Finn, Donal P.; and Brophy, Barry, "A Real Time Control Strategy for Optimisation of anEconomised Indirect MultiTemperature Transport Refrigeration System" (2010). International Refrigeration and Air Conditioning Conference.Paper 1093.

[3]. Finn, Donal Patrick; Cabello-Portoles, Andreas; Smyth, Shane; and Brophy, Barry, "Evaluation Of Defrost Options For SecondaryCoolants In Multitemperature Indirect Transport Refrigeration: Mathematical Modelling & Sensitivity Analysis" (2012). International Refrigeration and Air Conditioning Conference. Paper 1313.

[4]. Winkler, Jonathan Michael; Aute, Vikrant; Radermacher, Reinhard; and Shapiro, Doron, "Simulation and Validation of a R404A/CO2Cascade Refrigeration System" (2008). International Refrigeration and Air Conditioning Conference.Paper 942.

[5]. Yamasaki, Haruhisa; Yamanaka, Masaji; Matsumoto, Kenzo; and Shimada, Gaku, "Introduction of Transcritical Refrigeration CycleUtilizing CO2 as Working Fluid" (2004). International Compressor Engineering Conference.Paper 1632.

[6]. Pfafferott, T. and Schmitz, G., "Modelling And Simulation Of Refrigeration Systems With The Natural Refrigerant CO2" (2002).International Refrigeration and Air Conditioning Conference.Paper 577.