Prediction of HAVC Cool-Down Performance inside a Minibus Passenger Cabin Using CFD and Its Experimental Validation

Nikhil Mhetre[*1], Suraj Sathyanarayan[2], Manoj Diwan[3], Siddharth Kumar[4] and Dattatray Hulwan[5]

1.* Research Scholar, Department of Mechanical Engineering, Vishwakarma Institute of Technology, Pune, India. e-mail : nikhil.mhetre17@vit.edu
2,3,4 DGM R&D, Sr. Manager R&D, Team Lead CAE, MAHLE ANAND Thermal Systems Private Limited, Chakan, India.
5. Associate Professor, Department of Mechanical Engineering, Vishwakarma Institute of Technology, Pune, India. e-mail : dattatray.hulwan@vit.edu

Publication Info

Article history :

Received : 20th June, 2019

Accepted : 20th July, 2019

DOI : 10.18090/samriddhi.v11i01.7

Keywords : Cool-down, CFD, HVAC, Thermal comfort, Wind tunnel.

***Corresponding author :**

Nikhil Mhetre

e-mail : nikhil.mhetre17@vit.edu

Abstract

Now with more time spent by people while travelling and increasing mobility, providing passengers with a thermally comfortable experience are one of the important targets of any bus manufacturer. Conversely, comprehensive assessment through Climatic Wind Tunnel testing is costly and not possible during early stages of vehicle design. The aim of this work has been to develop a simplified simulation methodology to model the Minibus passenger cabin for cool down test. This study presents a methodology for predicting Heating, Ventilation and Air Conditioning (HVAC) cool-down performance inside Minibus cabin using Computational Fluid Dynamics (CFD) simulation to revise the HVAC duct design and parametric optimization in order to ensure thermal comfort of occupant. Heat Load is calculated analytically and has been considered in the CFD model and occupant heat load is considered as per ASHRAE standard. CFD simulation predicted the temperature and velocity distribution inside passenger cabin. Simulated cool-down results were found to be in good agreement with the experimental results. CFD cool-down prediction is useful in order to reduce time and costs related to climatic wind tunnel and road tests. Validated CFD model is used to study the effect of air flow on cool-down performance.

1. INTRODUCTION

The Minibus Segment is nowadays popular in Indian bus market. As per AIS 052:V4 Code Type IV bus is termed as Minibus. It is to be designed, constructed for occupant capacity between 8 to 16 and driver. Mini-Buses are distinct from passenger cars, having large glass area, large cabin volume and large occupant density per unit volume.

Changing global climate is leading to increased demand for air conditioning. On the one hand due to increasing demand of aesthetics bus manufacturers are adopting larger glass area. The intended reduction of the aerodynamic-drag resistance leads to body design with strongly inclined and therefore larger front windscreen. Due to this the total amount of solar heat loads into passenger cabin through the glass is increased substantially. This makes it challenging to satisfy subjective thermal comfort as per ASHRAE [1]. Thermal comfort issue is observed due to large variations inside passenger cabin.

Total Heat load for minibus ranges from 9kw to 14kw depending upon various design factors such as glass area, glass inclination, seating capacity and engine capacity [2]. The summation of all loads encountered by cabin is total heat load of vehicle. This total heat load should be taken away by HVAC system to keep cabin within thermal comfort conditions. Total Heat load is summation of solar radiations coming through glass, engine heat through firewall, exhaust heat through floor, ambient load through sheet metal body panels and heat rejected by occupants due to metabolic activities [3]. The cabin air interacts with total heat load and it causes to rise in the in-cab temperature. The passenger cabin is insulated with poor thermally conductive material to maintain low temperature for long run.

Development of good HVAC system is essential in buses, given that they are widely used for mass transportation.

2. LITERATURE SURVEY

There are various methodologies published in literatures to evaluate cool-down performance and thermal comfort.

Mhetre N. et al [2] estimated detailed heat load for minibus passenger cabin and studied variation of solar radiation load with respect to time. Yinhua Z, et al [3] developed method to estimate heat load and validated in the wind tunnel testing. It is much accurate in calculating OSA modes than that in REC mode. The calculation performed in forms of simple, fast and accurate manners by just entering the calculated vehicle geometry and weather conditions. Pawar, S. et al [4] established a suitable use of CFD for the assessment of passenger cabin climate control system in a city bus during the development phase. A structured methodology was used wherein important parameters like airflow rate, mean age of air (MAA) and human thermal comfort module were effectively used to gain insight for evaluating cabin climate control conditions and optimizing the air delivery system for the bus cabin. They have good agreement with the test results, confirmed the validity and reliability of the current CFD model. Daithankar, N. et al [5] developed methodology for predicting thermal comfort inside Midibus cabin and is validated very well with on road experimental datas. This methodology is helpful for predicting occupant thermal comfort with the given HVAC system in the hot climate conditions and showed that it meets the thermal comfort requirements of the passengers. They observed that the incident solar radiation through glass inside passenger cabin is higher which leads to increase in temperature of passenger cabin. Piovano, A. et al [6] developed an advanced methodology to predict the cabin cool-down test of car. They tried to catch the correct heat transfer between the outside environment and the internal cabin with a thermal tool, together with an internal flows CFD simulation by focusing the attention on the fluid dynamics and thermo dynamics aspects of the cool-down physics. They completed 30 minutes of cool down simulation in 5 days, on 112 CPUs for CFD runs.

The literature offers many studies that analyze the cabin climate conditions inside car and bus. These simulations can take a huge amount of computational resources and time, and requires exact geometrical data of vehicle exterior which is not available in early stages of vehicle design. [6] This simulation method, tested in environments characterized by non-uniform and variable temperature conditions on road test or static test, is not valid for HVAC cool-down performance validation as per industry standards. Some researchers performed steady state simulation which cannot represent actual case. [4, 5]

Therefore it has been decided to develop simplified and time saving methodology to predict cool-down Climatic Wind Tunnel (CWT) testing performance in CFD.

3. VCRS IN AUTOMOBILES

Vapor compression refrigeration systems (VCRS) are the most widely used in refrigerators, domestic ACs and automobiles. Automotive HVAC system consists of compressor driven by

S-JPSET : Vol. 11, Issue 1, ISSN : 2229-7111 (Print) and ISSN : 2454-5767 (Online)

engine pulley, condenser, Thermostatic Expansion Valve (TXV), Evaporator, blower and ducts. Evaporator is located inside space to be conditioned. Condenser is located at front side of engine bay before radiator. Refrigerant is normally R134a or R1234yf. The VCRS cycle is shown in Fig.1. The working fluid (refrigerant) undergoes phase change during condensation and evaporation. In a VCRS, refrigeration effect is obtained as the liquid refrigerant evaporates at low temperature in evaporator by absorbing heat carried by air present in cabin. After evaporator, refrigerant enters into the compressor as a saturated vapor and is compressed to a higher pressure and high temperature. The superheated compressed vapor is condensed with cooling air flowing across the tubes of condenser as refrigerant rejects heat and is carried away by air. The actual vapor compression cycle is called as Reverse Rankine cycle.

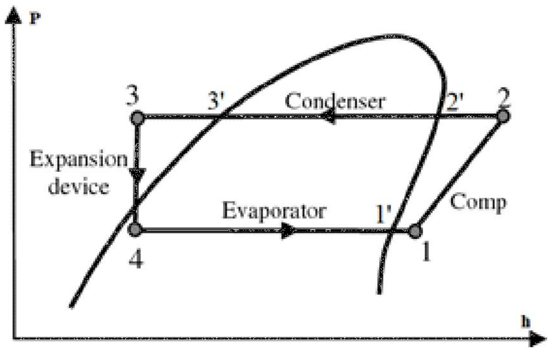

Fig.1: VCRS Cycle

3.1 Bus HVAC Architecture

The minibus segment is to be constructed for the purpose of transportation of passenger capacity varies between 8 to 16 plus driver and attendant. Every occupant should be within thermal comfort conditions. In order to provide uniform distribution of temperature inside passenger cabin three evaporators are connected in parallel conection. The locations of evaporators are shown in Fig.2 below.

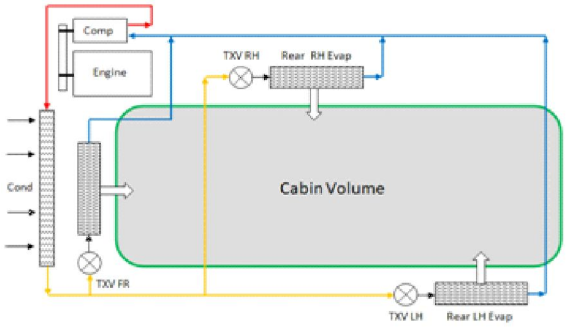

Fig.2: HVAC System Schematic

Total heat load is taken away by three units, two at rear and one at front of cabin. Two identical roof units are mounted on roof longitudinally. Front HVAC is mounted on firewall under dashboard.

3.2 *Experimental Test in Climatic Wind Tunnel*

The experiment performed on baseline vehicle in the CWT to get boundary conditions, required in simulation. It has been decided to perform the soaking and cool down test early in the program activity, in order to get all the boundary conditions data required by the simulation, such as the temperature curve of the air coming from grills, in-cab temperatures, nose level temperatures, blower inlet, evaporator outlet, sheet metal temperatures at roof, firewall, floor, side panel, rear door, the value of solar radiation intensity and air temperature in the tunnel during soak. Before the test the vehicle has been instrumented with all the thermocouples, voltmeter, humidity sensor and pressure gauges used in standard cabin cool down tests, plus additional ones useful for a detailed correlation analysis. The thermo sensors used in the experiment are K-Type thermocouples, with an accuracy of ± 1 °C, at the 95% confidence level.

The cool-down performance is evaluated on time based average Nose temperature curve. Nose temperature is measured at nose level of every manikin seating location with respect to time. The curve of average nose temperatures is examined to evaluate cool-down performance. Target is to achieve predetermined average nose temperature within specified time.

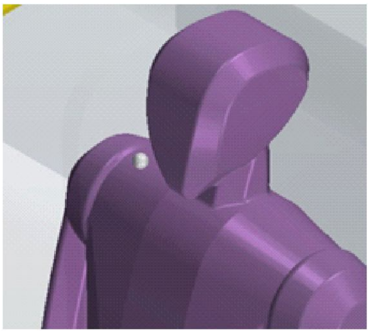

Fig.3: Location of Nose Thermocouple

The vehicle is soaked in the climatic test chamber for two hours before cool down starts. The vehicle is soaked under predetermined standard thermal loads. Soaking is performed in two phases, initially (Phase 1) all windows and doors are opened to get faster increase in cabin temperatures. In phase 2, all window and doors are closed to get faster soaking. Once the average cabin reaches the set point temperature the HVAC system is turned on with the blower in full speed and with the AC control settings as recirculation mode and face mode.

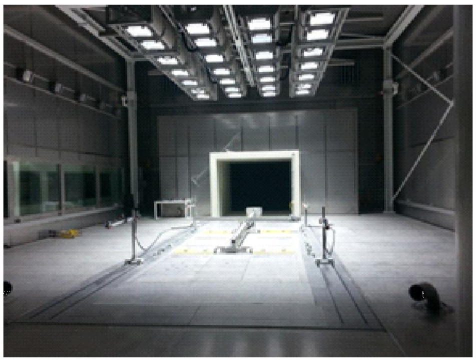

Fig.4: Climatic Wind Tunnel

A passenger cabin cool down test is usually composed of different phases corresponding to a typical drive cycle. The correlation activity focused on vehicle soak and on the first cool down phase of 30 minutes, performed at fixed drive cell conditions. The climatic wind tunnel environmental conditions were: air temperature 48 deg °C, relative humidity 50%, vehicle speed 40 km/h and solar array load 1200 W/m^2. Required data recorded at each second by using high speed data logger. The performance of the HVAC cool-down is then analyzed based on the average cabin temperatures at pre-defined time points.

3.3 CFD Simulation

The CFD simulations have been performed with commercial CFD tool STARCCM+ with transient-state computations and segregated solver. The HVAC cool-down performance is predicted by adopting a methodology shown in Fig. 4 below.

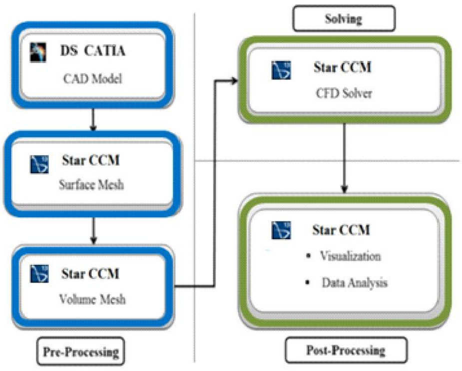

Fig.5: CFD Methodology

3.4 Pre-Processing

The pre-processing includes geometry modeling, mathematical modeling and meshing. In order to perform interior HVAC analysis, air cavity has been extracted from CAD model. The exterior parts like lamps, mirror, wiper, luggage compartment, superstructure chassis parts, engine, tires, suspension, etc. along with interior parts like wiring harness, seat adjuster, recliner, seat hand rest, speaker audio assembly etc. have been removed from CAD model. Further the CAD geometry is cleaned-up for any holes, sharp corners, duplicate surfaces etc. HVAC units are not considered in domain to avoid complexity and reduce simulation time. Required boundary conditions were applied at HVAC interface by performing experimental tests on baseline model.

3.5 Meshing

A mesh is a discretization of domain of interest into set of elements. The trimmed cell structured mesh is used to mesh all domains shown in Fig. 6. A finer mesh is generated in areas where the geometry has a large influence on flow and where large velocity and pressure gradient assumed to occur. To capture near wall region phenomena three prism layers are provided to domain. Fine volumetric control is provided around thermo sensor to capture exact flow physics as shown in Fig. 7. The total trimmed mesh resolution is 11 million.

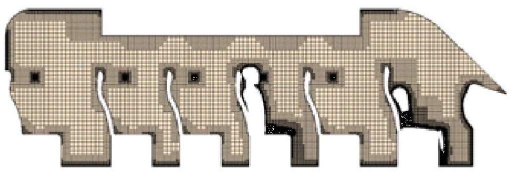

Fig.6: Mesh Scene at midplane

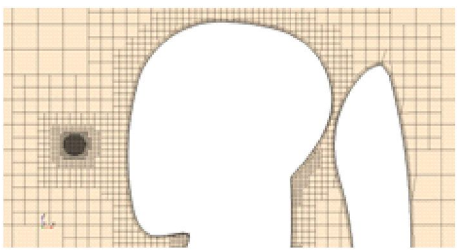

Fig.7: Volume Control near sensor

3.6 CFD Simulation Case Setup

Worst case scenario of thermal environment inside Minibus during peak summer season is considered. The standard k-epsilon turbulence model is used to account for turbulence effect. In computational domain the Yplus range is around min 0 to max 5. The standard wall functions with near wall effects is used to capture fluid behavior adjacent to wall.

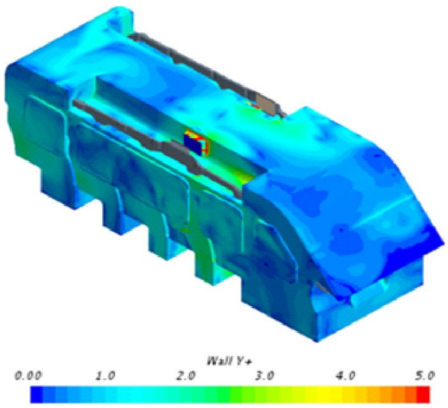

Fig.8: Domain Wall Y+

3.7 Boundary Conditions

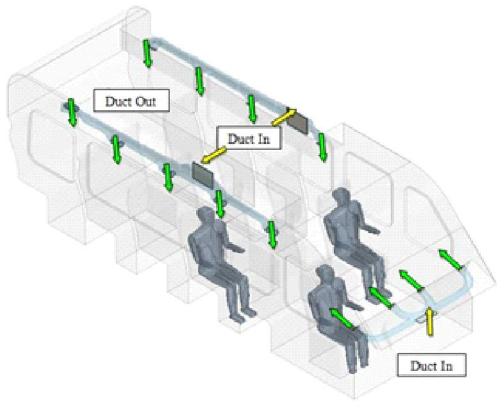

Fig.9: Boundary Conditions

Above Fig. shows visualization of CFD domain & boundary conditions for Cool down simulation. Here, we considered mass flow inlet at duct inlet and atmospheric pressure outlet at air inlet of HVAC. All boundary conditions are applied considering experimental data of baseline model. The average air temperature of evaporator outlet recorded in the experiment, has been used as temperature curve for the duct temperature to exclude HVAC units from the CFD domain, simplifying the case. The effect of heat ingress through sheet metal insulation and glass is calculated analytically and calculated heat load values are applied at respective boundaries, to reduce setup and computational time by excluding sheet metal insulation and glass from CFD domain.

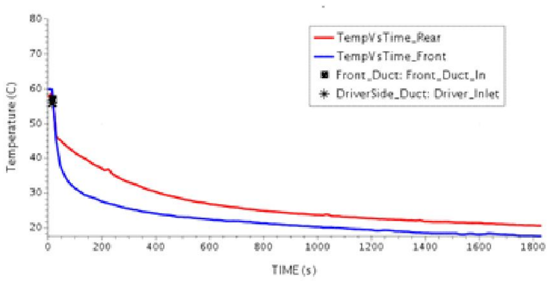

Fig.10: Evaporator out air temperatures

3.8 Simulation Results

In this paragraph the results of the transient cool down simulation are presented. The temperature curves show 30 minutes of cool down after soak phase. Fig. 11, 12, 13, 14 and 15 shows temperature distribution in 1, 5, 10, 20, 30 minutes of cool down.

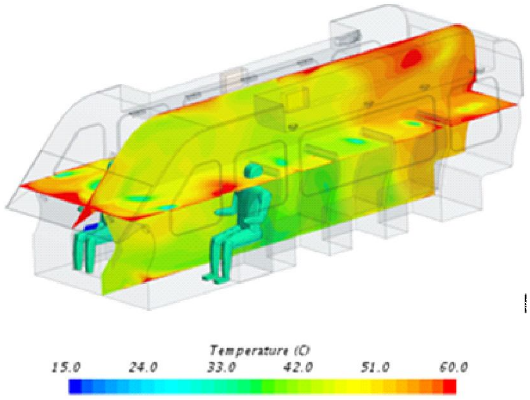

Fig.11: Temperature Distribution at 1min

Fig. 11 shows that the temperature distribution of cabin at 1th min is non-uniform. Mixing of low temperature conditioned air into initial hot air causing exchange of heat and resulting in reduction of overall temperature.

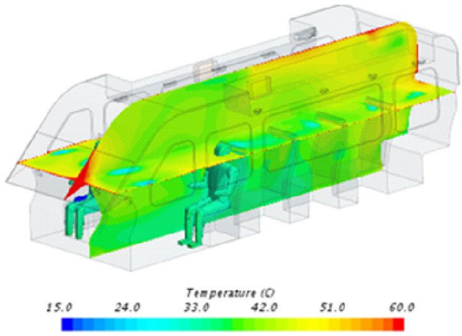

Fig.12: Temperature Distribution at 5 min

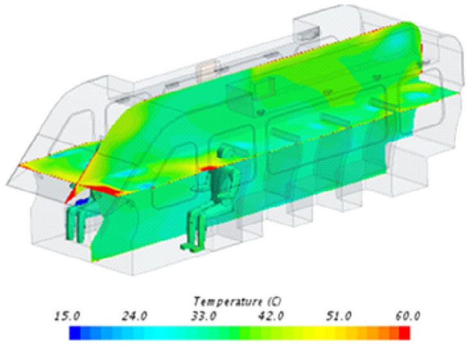

Fig.13: Temperature Distribution at 10 min

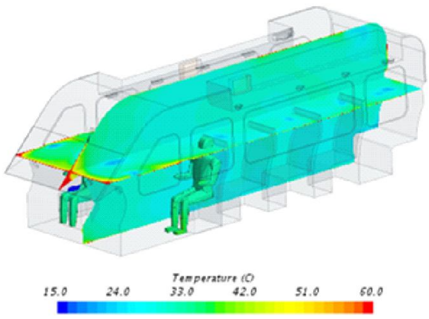

Fig.14: Temperature Distribution at 20 min

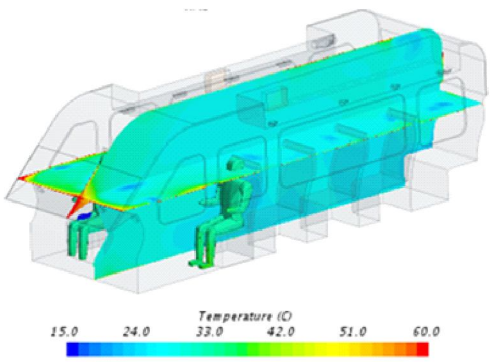

Fig.15: Temperature Distribution at 30 min

Fig. 15 shows that the temperature distribution of cabin in 30th min is uniform within range of 27°C to 29°C. It is observed that faster reduction in in-cab temperatures for first five minutes and slower drop later on. It is observed that duct is at lower temperature since it carries the conditioned chilled air. Maximum temperature is observed near front wind screen due to lower velocity and higher incident solar irradiation. The lower temperature is observed in normal direction of duct opening.

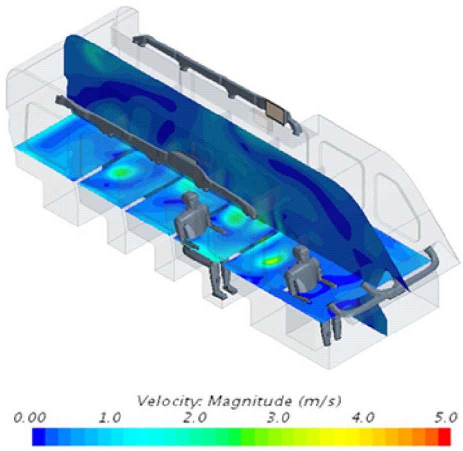

Fig.16: Velocity Plots at Mid Planes of Cabin

Fig. 16 shows the velocity plots at mid planes of cabin. It is observed that velocities are higher exactly near to grill outlet and reduce as move away from grill outlet. Velocities near nose are between 0.5 m/s to 1.5 m/s.

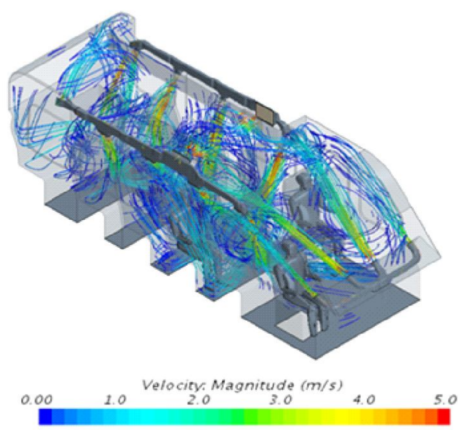

Fig.17: Streamline Plot inside Cabin

Fig. 17 shows streamline plot inside cabin, it is observed that streamlines from grills are covering maximum seating area and show better mixing with in-cab air.

4. RESULTS AND DISCUSSION

In this paragraph, the results of the transient cool down are presented and the validation is discussed. The temperature trends on simulated nose thermocouples are compared with the experimental ones. The temperatures profile show 30 minutes of cool-down after soak phase. All temperature values are in shown in Figures, both for experiment and simulation. It is observed that experimental results shows faster decay initially as doors are opened to occupy driver and operators before cool-down starts, resulting in some loss of in-cab heat which cannot be correlated in simulation.

Table-1: CFD and Experimental Avg Nose Temp

Time [min]	**Simulated Avg Nose Temp [°C]**	**Experimental Avg Nose Temp [°C]**	**% Deviation**
5	43.46	42.10	-3.22
10	38.10	37.30	-2.13
15	34.81	34.90	0.25
20	32.18	33.20	3.07
25	30.65	31.90	3.92
30	29.72	30.90	3.83

Table-1 shows the correlation of occupant average nose temperature and it is found that simulation results correlate well with experimental test within acceptable percentage deviation. It is observed that experimental nose temperature is higher than predicted due to infiltration effect which is modeled in simulation.

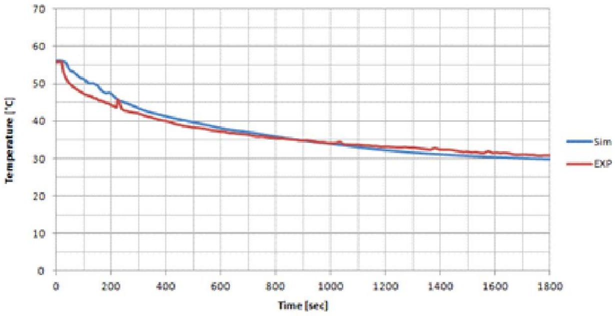

Fig.18: Simulation Vs Experimental Cool down Plot

Fig. 18 shows the correlation of simulation vs. experimental cool-down graph. It can be seen that the simulation results correlate very well with experimental cool-down results with very minimal deviation in the entire phase.

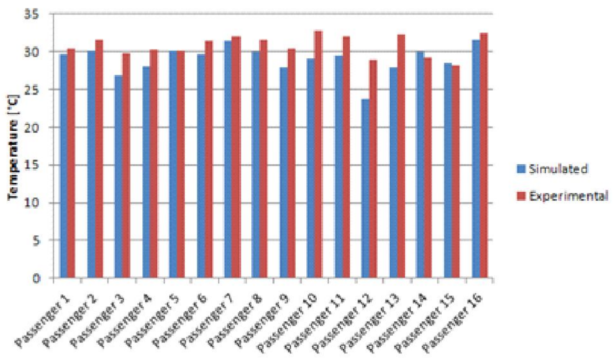

Fig.19: Nose temperature distribution (t=30min)

Fig. 19 shows the correlation of simulation vs. experimental nose level temperatures for each passenger location in 30th min of cool-down. It can be seen that the simulation results represent exact flow physics.

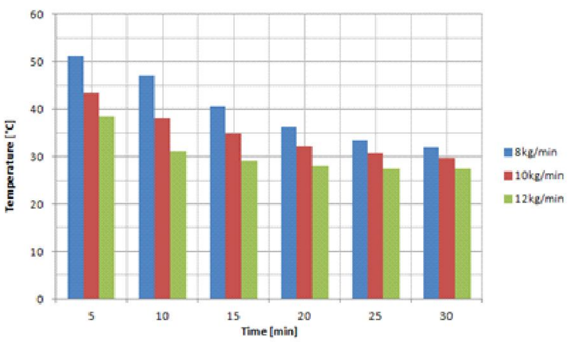

Fig.20: Parametric study based on air flow

Fig. 20 shows parametric study based on validated CFD cool-down simulation to predict effect of air flow per HVAC unit on cool-down performance. It is observed that with higher flow rate the cool-down temperature at predetermined time is reduced. Cool-down temperature is 27.4°C at 12kg/min and 31.8°C at 8kg/min.

5. CONCLUSIONS

- A simulation methodology has been developed for predicting HVAC cool-down performance for Minibus cabin and validated well with experimental test.
- The door opening and infiltration in the experiment is a noise factors that results in cabin air stratification, which cannot be correlated.
- Non-uniform temperature distributions, obtained especially in first 5 min of cooling period due to highly transient conditions, were occurred in 5-10 min of cooling period. On the other hand, the steady-state conditions were reached after 20 min of cooling period in terms of temperature distributions.
- Experimental Cool-down avg. nose temperature in 30th min is 30.9°C which can be optimized by increasing mass flow rate of air.
- Simulated subjective nose temperatures represent exact behavior and create base for future prediction of subjective thermal comfort.
- Model highlighted the seating locations where there is a need for flow improvement in order to achieve desired occupant thermal comfort.
- Effect of air flow on cool-down performance is studied based on validated CFD model. It is observed that with flow rate of 12kg/min, target of 28°C can be achieved.
- The presented methodology can be used in initial phase of vehicle design to reduce cost associated with number of expensive CWT and

road tests, and allow virtually analyze parametric studies. Air flow, grills sizing and positioning, duct design solutions for air distribution, are some of the key parameters that can be optimized with this model.

ABBREVIATIONS

HVAC - Heating, Ventilation and Air Conditioning

CFD- Computational Fluid Dynamics

PMV - Predicted Mean Vote

PPD - Predicted Percentage of Dissatisfied.

ASHRAE - American Society of Heating, Refrigerating and Air-Conditioning Engineers

AIS - Automotive Industry Standards

CWT- Climatic Wind Tunnel

RH - Relative Humidity

AVG - Average

EXP - Experimental

ACKNOWLEDGMENT

The authors would like to express their gratitude to Mr. Nikhil Desai and Mr. Pankaj Damle (Manager CFD) of MAHLE ANAND Thermal Systems Private Limited, Chakan, Maharashtra, India for their valuable technical suggestions, continuous supports.

REFERENCES

[1] ASHRAE Handbook of Fundamental, American Society of Heating, Refrigerating, and Air Conditioning, Atlanta, GA,1988.

[2] Mhetre, N., Sathyanarayan S., Diwan M., Hulwan D., , "Detailed Analysis on Air Conditioning Heat Load of a Minibus Passenger Cabin Considering Radiation Effect", International Journal Of Research And Analytical Reviews (IJRAR), may 2019, vol 6, issue 2, doi: 10.6084/m9.doi.one.IJRAR19L1177

[3] Yinhua Z., Brown M., and Harry Y., "A Simple Method to Calculate Vehicle Heat Load", SAE International 2011-01-0127, doi:10.4271/2011-01-0127

[4] Pawar, S., Gade, U., Dixit, A., Tadigadapa, S. et al., "Evaluation of Cabin Comfort in Air Conditioned Buses Using CFD," SAE Technical Paper 2014-01-0699, 2014, doi:10.4271/2014-01-0699.

[5] Daithankar, N., Udawant, K., and Karanth, N., "Prediction of Thermal Comfort Inside a Midibus Passenger Cabin Using CFD and Its Experimental Validation," SAE Technical Paper 2015-26-0210, 2015, doi:10.4271/2015-26-0210.

[6] Piovano, A., Lorefice, L., and Scantamburlo, G., "Modelling of Car Cabin Thermal Behaviour during Cool Down, Using an Advanced CFD/Thermal Approach," SAE Technical Paper 2016-01-0213, 2016, doi:10.4271/2016-01-0213.