A real time numerical analysis of vehicle cool-down performance

S HONG ROK
Hyundai Motor Company, Korea
H PASTOHR
P+Z Engineering, Germany

ABSTRACT

The cool-down performance of a passenger vehicle is affected by several components for instance the air-conditioning system itself, engine compartment and environmental thermal conditions. Therefore sufficient information is required at very early stage of design to ensure sufficient air conditioning performance for the passengers. In this study a robust numerical method for estimating vehicle cool-down performance developed recently is introduced. A full air-conditioning system was simulated by a commercial 1-D thermohydraulic analysis code, FLOWMASTER. Heat flux including solar radiation through the car structure was estimated by CFD simulation, using STARCD and a 3-D heat transfer program, THESEUS_FE v1.0. These programs were coupled together like a real air-conditioning system which shares boundaries between different systems. All thermal conditions were defined exactly with a Climate Wind Tunnel (CWT) used for developing a real car. It was found that this method provided results very similar to the test results during the entire simulation process.

1 INTRODUCTION

Since 1928 when refrigerant R-12 was introduced air-conditioning (AC) systems have been installed in almost all passenger cars Passenger comfort is becoming a very important sales issue since vehicle supply ratio has increased and people spend more time on the road. The greatly increasing consumption of refrigerant R-12, R-134 and fossil fuel began to effect ozone depletion and global warming. Therefore, global organizations like EC/EPA/CARB are trying to control refrigerant leak and fuel consumption by enforcing tight regulations and laws and by creating increasing pressure on vehicle industries and HVAC system suppliers to develop more natural friendly refrigerants. In order to meet the standards of these environmental regulations, firms for vehicle production make great efforts to improve the efficiency of the AC system and simultaneously to develop a new refrigerant reducing vehicle thermal load by applying solar reflective glass or reinforced thermal insulations. In addition, while maximum performance has been the most important target in developing an AC system. The focus has moved to how quickly passenger thermal comfort is achieved. Virtual

---

manufacturing is now widely used in the early stages of developing a vehicle. HKMC is continually increasing research expenditure to strengthen the frontloading in developing crash, vehicle dynamics, aerodynamics, NVH and underhood cooling performance. However the application of virtual manufacturing is less popular in the cool-down or heating of the passenger compartment because the computational time and modeling are still very expansive and the complicated boundary conditions and reliability have not yet been sufficiently explored and verified.

In this study we would like to introduce a vehicle cool-down simulation method which has proven more efficient and more reliable. Initially, the cool-down performance is developed in the climate wind tunnel. Therefore, to apply exact simulation boundary conditions we measured the solar radiation and surface temperatures on the car body and the chamber walls in addition to the cabin temperature.

We selected FLOWMASTER for the AC system analysis, THESEUS_FE for the solar radiation and 3-D heat conduction through the structure and STARCD for convection heat transfer. Cool-down performance is determined by influences within the AC system, CWT conditions and air distribution inside the cabin. So the most important technique of real time cool-down simulation is that all 3 programs should share their boundaries and exchange boundary information as often as possible. In this study we tried to estimate the cool-down performance numerically by developing a special interface that couples the 3 programs in real time while preserving the CWT conditions and the vehicle structure in depth. The result was very rewarding and we expect this could be a very substantial tool indecision making on the cool-down performance in early stages of the development process. Following we shall introduce the simulation method in detail.

2 AC CYCLE SIMULATION

This section describes the modeling technique using the efficiency data of the A/C system configuration parts used in the vehicle. Figure 1 shows the layout of the A/C system affixed in the general passenger car quantity. In most passenger cars the A/C system is manufactured with tube backs connecting each component such as the Compressor, the Condenser, the Evaporator and the Expansion system. Compressor efficiency is evaluated by accounting for the compression ratio. It finds its expression in a formula which is a surface function of volumetric efficiency and isentropic efficiency. Figure 2 shows examples of the volumetric and isentropic efficiencies of a swash plate compressor. The test data should be sufficient to cover the entire driving conditions during testing.

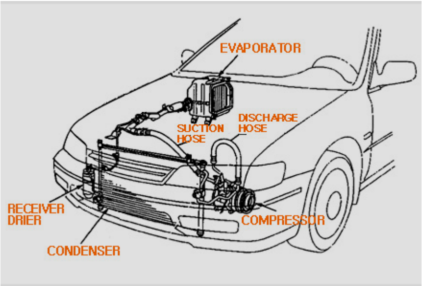


Fig. 1 AC system layout

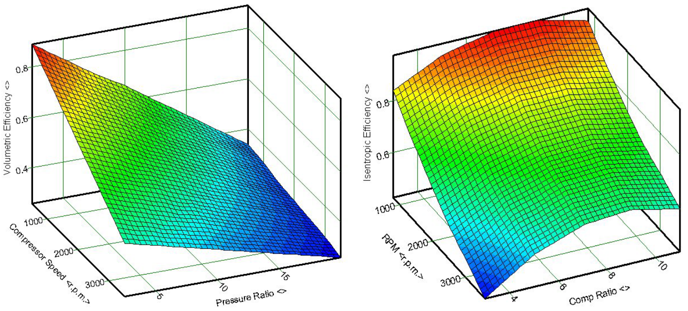
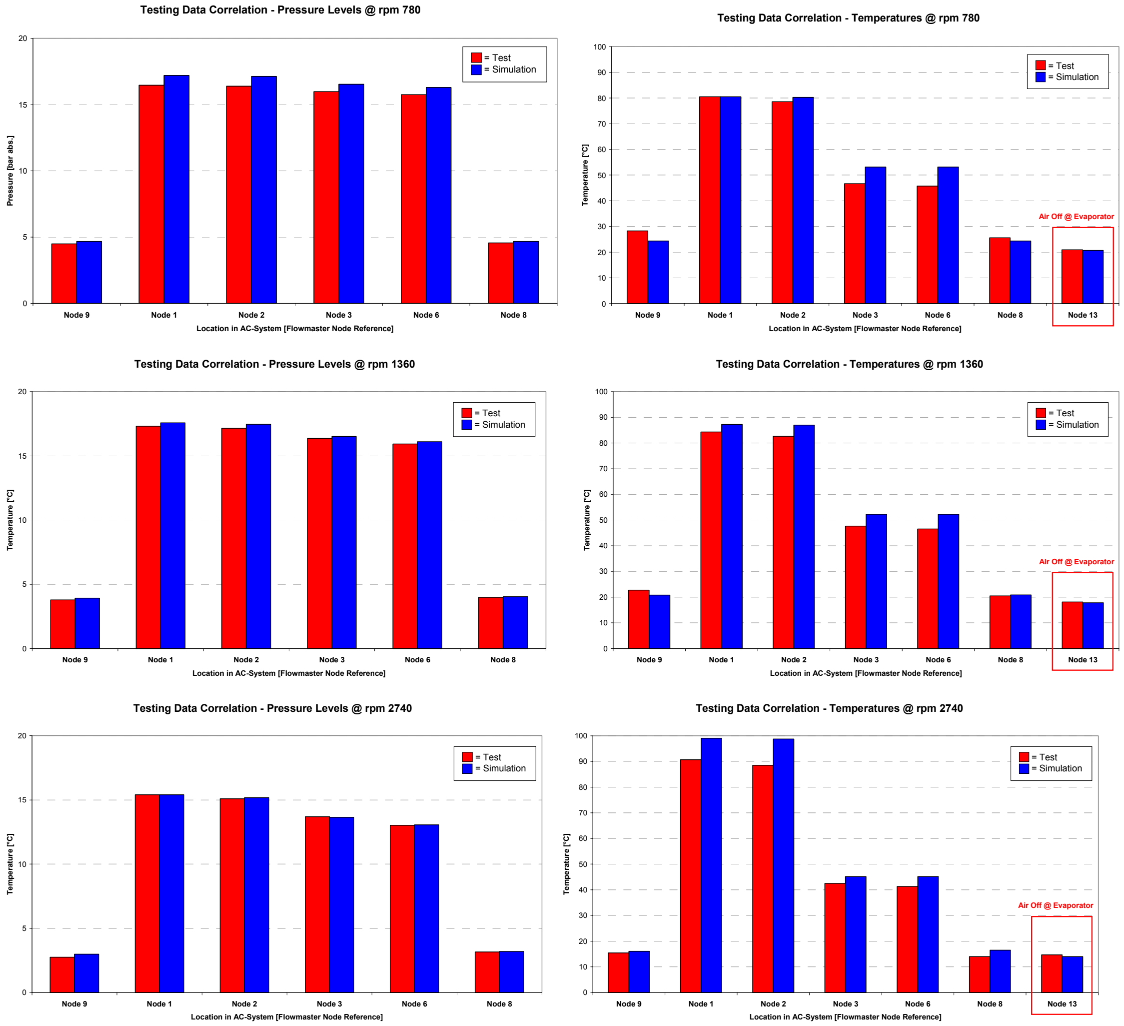


Fig. 2 Volumetric (L) and Isentropic (R) efficiency

The Condenser and the Evaporator are heat exchangers and especially the evaporator accompanies the phase change. The geometry is so complicated that it is very difficult to estimate the performance numerically. FLOWMASTER simulates the performance based on several test results and creates normalized characteristics. They are called shape factors and consist of air side heat transfer coefficient (HTC) and pressure drop (ΔP), refrigerant side HTC and ΔP and condensation mass flow rate on the Evaporate. The refrigerant flow rate is controlled by a Thermal Expansion Valve (TXV). We modeled it with the opening area of the nozzle versus superheat.

Table 1. Simulation conditions

Speed [rpm]	Condenser on velocity [m/s]	Evaporator on volume flow rate [m3/s]
780	1.0	0.14
1360	1.5	
2740	4.5	


Testing Data Correlation - Pressure Levels @ rpm 780

Testing Data Correlation - Temperatures @ rpm 780

Testing Data Correlation - Pressure Levels @ rpm 1360

Testing Data Correlation - Temperatures @ rpm 1360

Testing Data Correlation - Pressure Levels @ rpm 2740

Testing Data Correlation - Temperatures @ rpm 2740

node No.	1	2	3	6	8	9	13
Position	Comp. out	Cond. in	Cond. out	TXV in	Evap. out	Comp. in	Evap. air off

Fig. 3 Validation of the AC cycle simulation [1]

Driving conditions differ between car manufacturers. HKMC evaluates the performance at variable speeds and temperature depending on the sales countries. These evaluation conditions must be considered as simulation boundary conditions. The air flow rate on the Condenser was obtained from CFD simulation and that on the Evaporator was measured. Before coupling with other programs the AC system network was validated by comparing with the test results. The test was performed at low speed, high speed and idle conditions like Table 1. Figure 3 illustrates the results.

The simulation results were very similar in all driving conditions. Generally the temperature gives rise to larger errors than the pressure especially at node 1 and 2, the error growing with speed to a maximum of 11%. However, the air off temperature on the Evaporator is very accurate with less than 5% error. With these results we simulated transient cool-down by coupling 3 programs.

3 VEHICLE HEAT LOAD SIMULATION

In order to determine cooling capacity it is essential to calculate the vehicle thermal load under severe test conditions. The CWT is set up at 45℃ with 1000W/m^2 solar load. Before testing the test car had been kept standing for several hours until the car accumulated a predefined thermal load. This soak period determines the initial cool-down speed. Therefore, the cool-down simulation must include this period. Considering the car structure in detail it is necessary to determine the final soak condition exactly because the car structure composes of several different material properties. Details of each part which have several layers with different properties are considered in THESEUS_FE. We used two different meshes for soak and cool-down cycle each. For simulation efficiency, the soak mesh is coarser than the other because the air movement is slow and all equations should be solved at the same time. On the other hand, we can manage larger size mesh for the cool-down cycle which is characterised by a large amount of ventilation airflow. The momentum and energy equations are solved consecutively, the buoyancy effect was neglected [2].

Figure 4 shows a shell mesh and a volume mesh. The former one is used for THESEUS_FE which takes the radiation interaction between shell elements into consideration solves the conduction equation through different shell layers by using finite element method. The shell layers were not considered as solid but numerically. The meshes consist of quad and tria types and the numbers are 74000. The latter is used for STARCD which calculates the air velocity and temperature distribution by the convection heat transfer. The elements are all tetrahedral and the numbers are 1.5 million are required for the cool-down simulation and 0.6 million for the soak simulation.

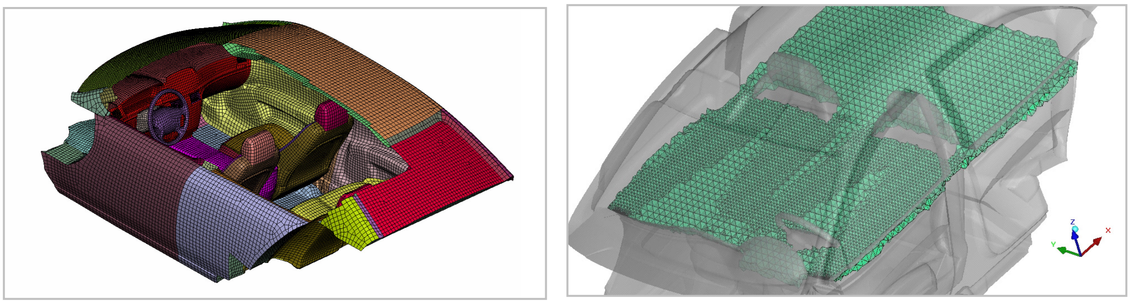


Fig. 4 Shell mesh (left) and volume mesh (right) models

The cool-down model does not include every detail such as the heater, blower, cross-member, harness and wiring which are installed behind the crash-pad. The thermal boundary conditions on the dashboard and floor are obtained from the CWT test. More details are described in the following section.

The total airflow rate supplied in the cabin by the HVAC system and the air distribution of each vent were taken by experimental measurement. In addition, we calculated the heat transfer coefficient (HTC) value on the surfaces of the car via the aerodynamic simulation of the full vehicle geometry to account for the heat transfer change for different driving speeds. The cooling airflow rates on the Condenser were obtained by under-hood simulation taking into consideration fan rotation and driving ram air flow.

4 COOL-DOWN TEST IN THE CWT

In order to obtain all the required thermal boundary conditions and the AC performance for correlation with the simulation results under pre-defined test conditions we carried out cool-down tests in the HMC CWT. The test was carried out at three different vehicle speeds - low, high and idle - respectively. Speed shift was controlled automatically by a robot to ensure data undisturbed by cabin passengers. The test shows that the air temperature surrounding the under-floor and the dash-board parts was about 60℃ during driving. The temperature picked up over 100℃ near the exhaust pipe when idling. We also found that the solar load recorded in the pyranometer which was installed inside the cabin decreased during driving. It is assumed that this phenomenon comes from the cooling effect of the solar light because the measuring device measures the full spectrum of the radiation. The wall temperatures of the CWT were about 50℃ even though the controlled air temperature in the test section was 45℃. The heat effect between the recirculation air inlet position and the evaporator inlet was trivial.

All information was applied as boundary conditions for simulation. Figure 5 shows the simulation boundary conditions.

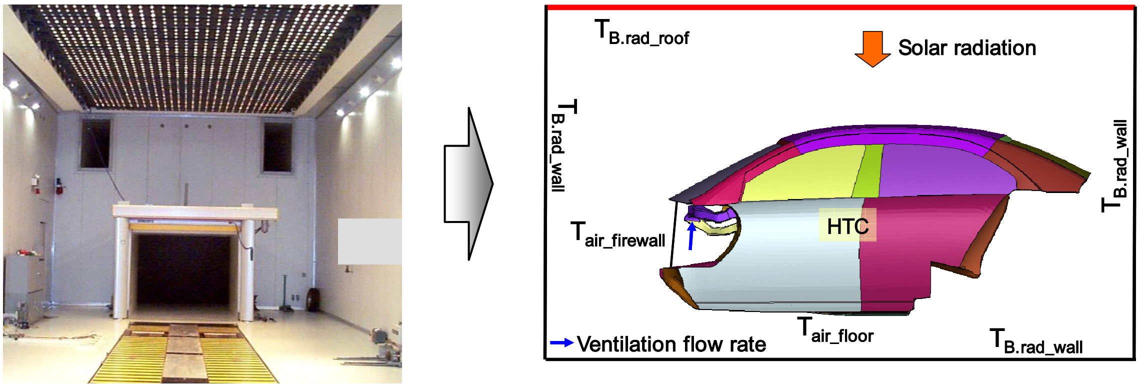


Fig. 5 Simulation conditions for the CWT test [1]

The cool-down performance of the test car was thus verified.

5 METHOD OF THESEUS_FE - STARCD - FLOWMASTER - LINK

The simulation programs are linked to solve the cooling process equation in an automotive car more accurately and realistically. The method is based on combining an (1D) AC-Network thermal analysis with a thermal and fluid dynamic model of the ducts and the cabin (2D shells and 3D volumes) under consideration of the inner and outer surfaces of all necessary parts [3].

THESEUS_FE is a thermal simulation program for solving radiation interaction between shells and the conduction inside shell layers by using the finite element method. The radiation model is based on a gray model. The radiation interaction is taken into consideration by a surface to surface radiation model. To solve the very large numbers of view factors in time a patching method is used.

STARCD is a simulation program for solving the Navier-Stokes equations of thermal fluid dynamics. Turbulence will be modeled by a High Reynolds turbulence model.

The temperature and concentration at the evaporator outlet (calculated by FLOWMASTER) is transported to the THESEUS_FE ducts. THESEUS_FE calculates the temperature increase of the ducts (heat loss) from the FLOWMASTER out put data, by assuming a 1D AIRZONE model. The dashboard volume temperature is linked via the convection in the dashboard to the duct temperature. The outlet values of the ducts (temperature and concentration) are transported to the inlet of the cabin and mathematically accounted for by STARCD. The flow field transports the temperature and the concentration from the inlet vents of the cabin to the outlet (HVAC Blower for recirculation). On the outlet (pressure outlets) the temperature and concentration will be averaged and sent to the evaporator inlet of FLOWMASTER.

The cabin conditions will be accounted for with the boundary conditions of the walls temperatures. The temperature will be updated by THESEUS_FE at the beginning of each STARCD time step. STARCD writes out two values, the averaged fluid temperature and the averaged heat transfer coefficient (HTC), on all linked STARCD wall boundaries for each THESEUS_FE shell element. THESEUS_FE calculates the convection heat flux for each THESEUS_FE element with help of the fluid temperature and the wall temperature. The shell mesh of THESEUS_FE and the shells of the STARCD volume mesh are not geometrically connected. This gives the user a high flexibility in the project work.

The vector data (HTC, element temperature, fluid temperature and concentration) and the scalar data (averaged temperature and concentration on the outlet of STARCD, data from evaporator outlet, the next time step size, maximum time to solve, maximum THESEUS_FE time step and a flag if STARCD has converged) was moved between the different programs with a client server program programmed by P+Z Engineering.

A special library was developed to calculate all necessary data and transfer them via the network. The library is linked at runtime with STARCD and THESEUS_FE. The information for each STARCD property (shell) linked with a THESEUS_FE property is pre-calculated by a macro in PROAM/PROSTAR. During calculation this information will be used to find each THESEUS_FE element, the STARCD boundary and the volume attached to them.

The foremost task of the library is to read and check the input files written by PROSTAR. Then a subroutine searches for baffles and for STARCD boundaries. With the help of the temperatures and mass flux from THESEUS_FE the library can apply the values for each STARCD boundary. At the end of a converged time step STARCD writes out the HTC, fluid temperature and concentration. The library is programmed in serial and parallel to find the correct STARCD values (by using MPI for parallel).

Figure 6 describes the simulation flow in detail. A special controller can manage the time steps of STARCD and THESEUS_FE during the calculation interactively. The link simulation coupling different shell mesh size and type allows more flexibility to users concerning simulation time and limitation of hardware. It is very difficult to build a numerical model for cool-down simulation of a passenger car because of the complicated geometry. Therefore, simplification is usually necessary. However the missing parts also leave room for applying more realistic boundary conditions. These realistic boundary conditions help improve the reliability of the simulation. The same conditions can be applied for a similar type of car.

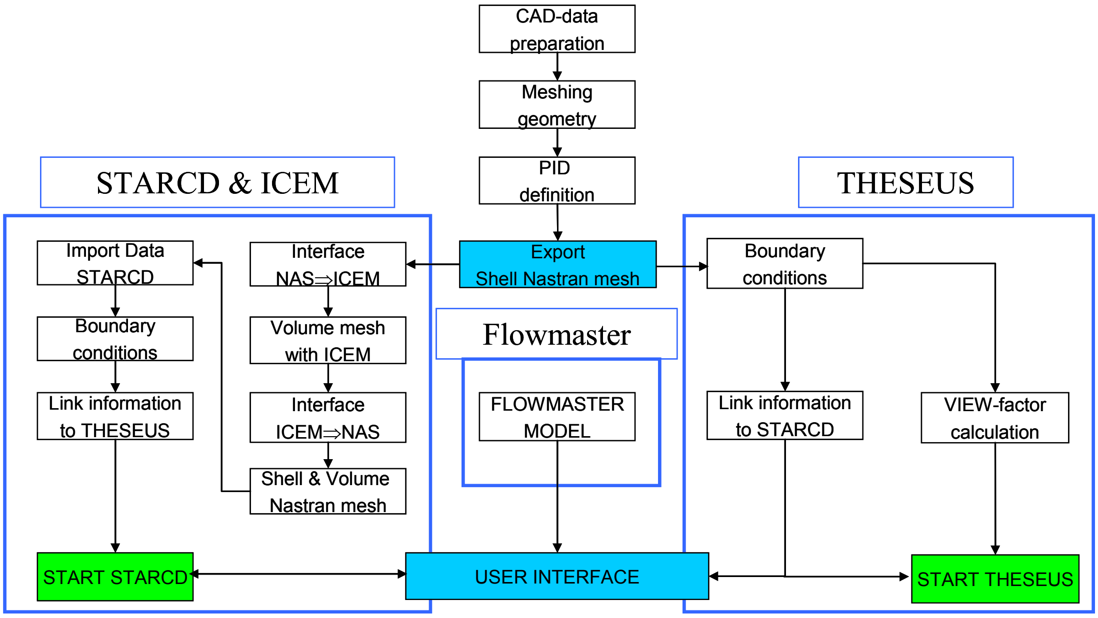


Fig. 6 Simulation flowchart

6 SIMULATION RESULTS

The cool-down simulation was performed for 5400 seconds not accounting for the soak cycle. The time step was gradually increased to 10 seconds for STARCD and 30 seconds for THESEUS-FE. FLOWMASTER and THESEUS-FE was run using Window NT, on an hp xw8000 and STARCD was run on AIX v5 parallel processor machine running under Linux.

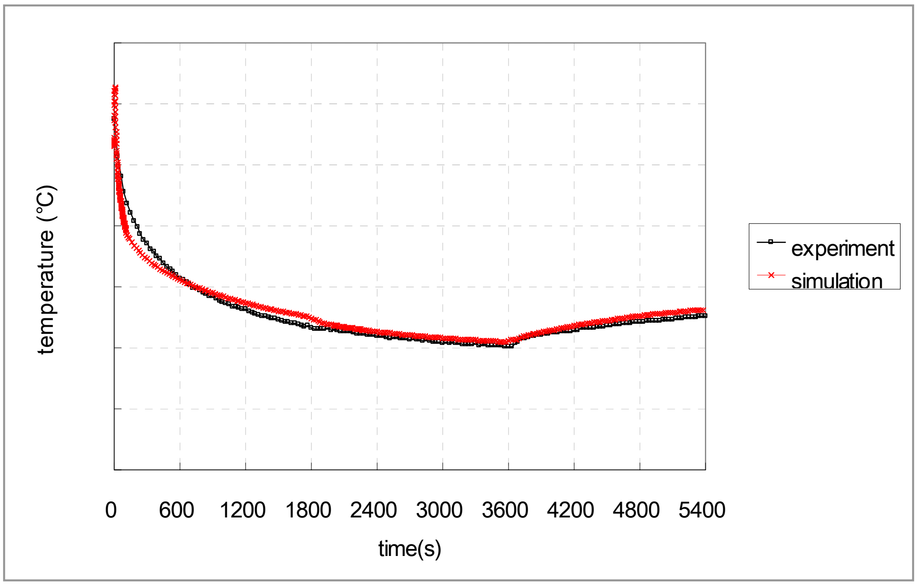


Fig. 7 Validation of the cool down simulation

Figure 7 shows the average cabin temperatures. At beginning of the test the simulation showed the cabin cooling faster than the experiment. After 10 minutes the cooling rate was similar to that of the experiment. The maximum error was around 1.6℃ after 30 minutes and 1.0℃ at the end of testing. The experiment result is the average value of 32 measuring points and the simulation data were taken from the same positions. The fast cool down speed at the beginning comes from FLOWMASTER because it cannot simulate the initial transients' condition of the real AC system correctly.

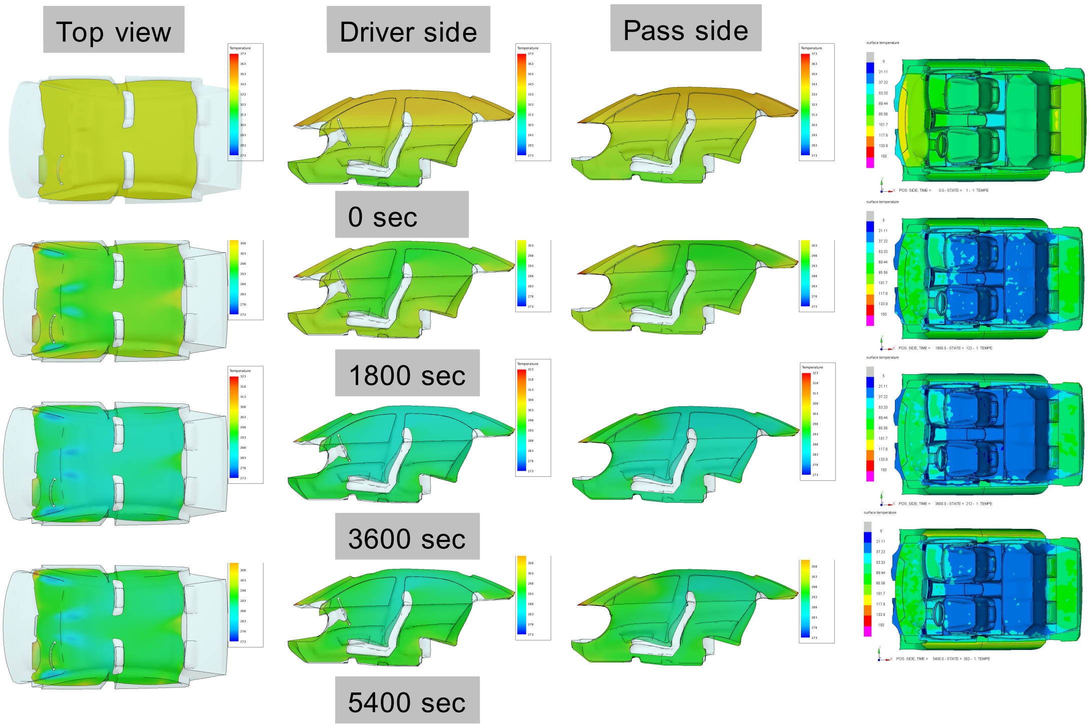


Fig. 8 Air (left) and surface (right) temperatures in the cabin

Figure 8 shows the air temperature distributions and the interior surface temperatures. They show the temperature stratification at the end of the soak due to the buoyancy effects. The cabin air temperature decreases until 3600 seconds have passed and often increases for the last 1800 seconds depending on driving conditions. The surface temperature is much higher than the air temperature. Especially the crash pad remains at a high temperature throughout the cool down. This affects not only the air temperature around the front passenger but also the passengers' thermal comfort.

7 DISCUSSION AND CONCLUSION

This study introduced a novel vehicle cool down simulation method which has been developed to account as closely as possible for real situations. It accounts for transient conditions by updating the operating conditions of the AC system, the ventilation air temperature and surface temperature of the vehicle structure. It calculates all energy transfer at the same time including solar and surface radiation, convection and conduction. We also considered the composite structure of different material properties.

To link all the programs which run on different operating systems we developed a special interface to transfer data via the network. This method establishes a strong connection on the border between the AC system, HVAC and cabin as in a real car. For simplification of modeling we applied the boundary conditions obtained from the CWT test.

Except for the initial cool down speed the simulation estimated the cabin temperature very well throughout the complete process. It also supplied complete information about the temperature and humidity concentration distributions in the cabin and the wall temperature of the interior. Furthermore it produced full data about the operating AC system. We expect this numerical method will be a very useful tool in developing AC systems at an early design stage for both supplier and OEMs alike.

Future work is concentrated on improving the initial excessive cool down phenomenon.

8 REFERENCES

(1) Internal test report of the vehicle cool down performance, Hyundai Motor Company R&D Division, 2006.

(2) THESUES-FE Theory Manual v1.0, P+Z Engineering GmbH, 2006.

(3) Internal research of Hyundai Motor Company R&D Division with the support of P+Z Engineering from 2005 to 2006