ANALYSIS OF FLOW PATTERNS AND HEAT TRANSFER IN GENERIC PASSENGER CAR MINI-ENVIRONMENT

Alex Alexandrov[1], Vladimir Kudriavtsev[2] and Marcelo Reggio[3]
alexandr@cerca.umontreal.ca, vvk@cfdcanada.com, marcello@cardia.meca.polymtl.ca
[1]Research Engineer, CFD Canada / CERCA, Montreal
[2]Managing Director, CFD Canada, Toronto
[3]Professor, Ecole Polytechnique / CERCA, Montréal

Key Words: CFD, heat transfer, simulator, HVAC

ABSTRACT

In this paper, we use two and three-dimensional computer simulations to address issues of climate control and performance of Heating, Ventilation and Aero-Conditioning (HVAC) system of a generic passenger car.

We examine the role of HVAC configuration and design parameters, such as air temperatures and velocities at the inlets, the size number and location of the system's inlets and outlets. We also examine impact of external parameters, such as outside temperature and vehicle velocity on climate conditions inside the car.

INTRODUCTION

High levels of vehicle comfort are being increasingly demanded by users. This creates a new challenge for climate control engineers. In the past, it typically took a year to develop and fully characterise the Heating, Ventilation and Aero-Conditioning (HVAC) system for a new model vehicle using conventional physical testing methods [1]. However, the use of computational fluid dynamics (CFD) simulators can dramatically reduce time of development of automotive HVAC systems, contribute to improvement of their performance and provide better understanding of the underlying processes [2], [3].

We conduct a set of computational experiments in order to isolate the most important parameters that affect car HVAC system performance and the impact of these parameters on the system. To perform the computations, the CFD-ACE+ [4], multidisciplinary CFD and heat transfer software, is used. With this package, we simultaneously solve internal-external flow problems with turbulence and heat transfer. We use the ?-? turbulence model because it is the most robust and is applicable to the widest range of problems.

SIMULATONS

We distinguish two categories of parameters, i.e. external and internal parameters. External parameters may include the velocity of the vehicle, air temperature outside the vehicle, the direction and velocity of the wind, etc. The second set of parameters, internal, may include the air temperature and velocity at the HVAC inlets, the number and location of inlets/outlets, thermo-conductive properties of the car walls, etc. Consequently, we are simulating a coupled problem with two components, being internal and external.

External Parameters

Car Velocity

The first observation arising from our simulations is that the vehicle velocity is of certain importance to cabin climate indicators, such as flow and temperature distributions. However, the influence of the vehicle velocity on the cabin climate decreases with increase of the velocity magnitude. This phenomenon is illustrated in figure 1.

Figure 1a presents a two-dimensional, steady-state case of heating of an immobile car. For figures 1b, c and d, the car is moving with three different velocities 5, 20 and 100 km/h respectively.

The other boundary conditions are listed below. The temperature of the air at the first inlet (located below the windshield) is 308K (35^0C). The horizontal and vertical components of air velocities at this inlet are U = 0.065m/s and V = 0.035m/s, respectively. The temperature at the second inlet (located on the dashboard) is 308K (35^0C). The air velocities at the second inlet are U = 0.5m/s and V = 0.05m/s. The only one outlet is placed below the rear window. Although the passenger temperature is equal to 309.65K (36^0C), the boundary conditions along the body have been selected according to experimental data in such a way that there is a very limited heat exchange between the body and the interior of the car. The conditions of the car walls represent thermo-conductive properties of the materials they are made of. The boundary conditions along the seats are considered to be adiabatic.

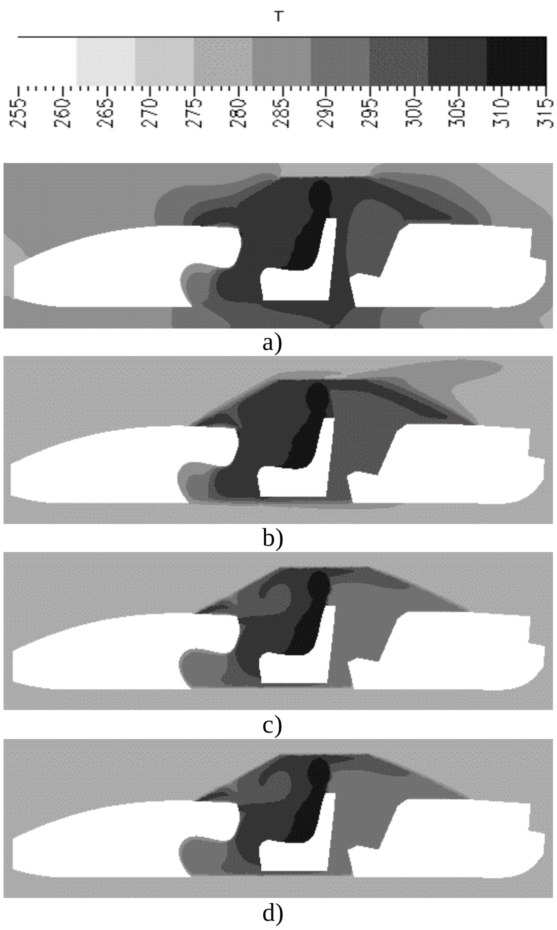


Figure 1. Temperature distributions for four different car velocities. U = 0, 5, 20 and 100 km/h

As it can be seen from the case of the immobile car (figure 1a), a cloud of warmer air is formed around the car, the wall temperature increases while the car interior temperature remains high. For the case of a slowly moving car (figure 1b), this warm cloud transforms into a warm trail behind the car and the interior temperature decreases. For the faster moving car (figures 1c and d), there is no significant difference in temperature distributions. This can be explained by the fact that with higher velocities the warm cloud around the car is dispersed and the temperatures of the external surfaces of the car walls are always equal to the outside temperature. Streamline patterns for the case of a fast moving car (conditions as figure 1d) are illustrated in figure2.

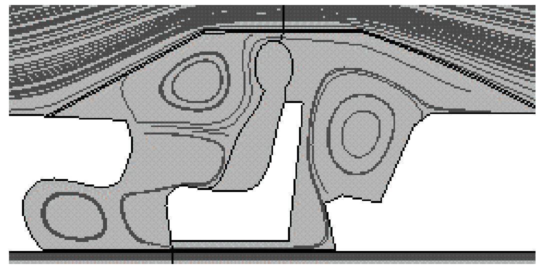


Figure 2. Streamline patterns inside and around the cabin

Outside Temperature

Another external parameter that significantly affects car HVAC system performance is the temperature outside the car.

For figure 3, we set all conditions the same as for figure 1d with exception to the outside temperature that is 268K (-5^{0}C), rather than 278K (5^{0}C). From this figure, it can be seen that for the case with a lower outside temperature, the car interior temperature is also lower.

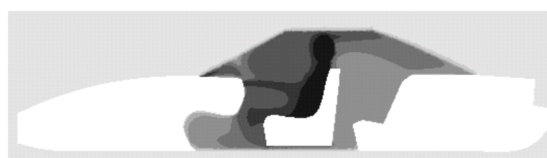


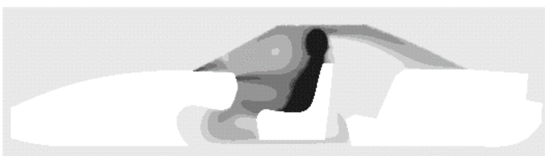
Figure 3. Steady state case with a lower outside temperature

Internal Parameters

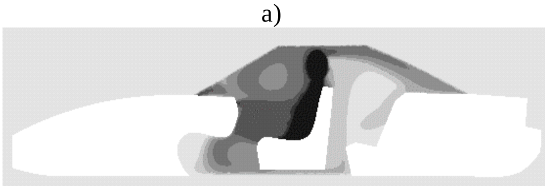
Inlet Conditions

Not surprisingly, both the air temperature and velocity at the inlets play an important role in determining cabin climate. Another observation is that within the realistic range of inlet velocities and temperatures, there is a constant presence of "cold" (in the case of heating) or "hot" (in the case of aero-conditioning) areas. This trend is more pronounced at the beginning of heating/cooling. Figure 4 illustrates this effect for a transient case of heating. The boundary conditions are the same as for the steady-state case with exception to the outside temperature which is equal to 263K (-10^{0}C) and the *U* component of the velocity at the second outlet which is higher and equal to 0.95m/s (higher velocities at the beginning is a better replication of the reality). The snapshots are taken at five time instants, i.e. after 20, 40, 80, 160 and 380 seconds.

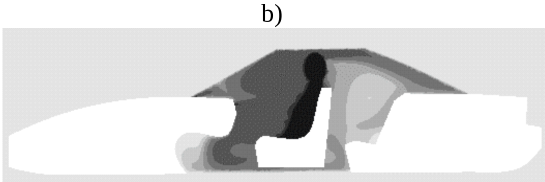
There are two main locations of the "cool" areas, being behind the front seats and in the front seat leg areas. These cool areas typically coincide with areas of slow air circulation as in figure 5. We expect that this effect can be slightly mitigated in three-dimensional cases. It is also our belief that this effect can be minimised by optimising inlet/outlet locations.



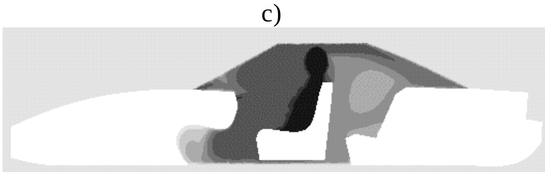
a)



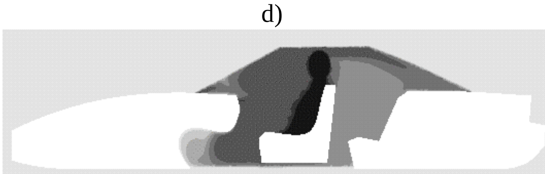
b)



c)



d)



e)

Figure 4. Transient heating of a car. t = 20, 40, 80, 160 and 380s.

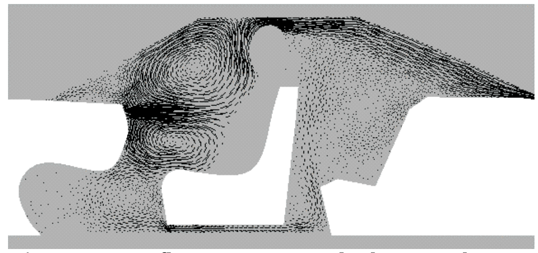


Figure 5. Air flow patterns inside the car cabin

Inlet/Outlet Locations

In order to establish the role of the size, number and location of HVAC system inlets and outlets, we ran three-dimensional simulations for various inlet-outlet configurations. Complete optimisation of a HVAC system configuration represents a formidable task and is beyond the scope of this paper. For this paper, however, we present several characteristic cases of HVAC inlet and outlet placement.

? *Case One*

There are three inlets in the cabin. The first, slot shaped (3 X 80 cm), inlet is located below the windshield. The second and third (both 10 X 5 cm) inlets are located symmetrically on the dashboard 15 cm away from the sidewalls. Two equally sized (7.5 X 3.5 cm) outlets are located below the rear window 30 cm away from the sidewalls.

? *Case Two*

There is the same windshield inlet plus four dashboard inlets. The dashboard inlets are equally sized (10 X 5 cm) and located symmetrically 7cm and 60cm away from the sidewalls. The outlets are the same as in the first case.

? *Case Three*

The set-up for the third case is the same as for the second case with exception to the outlets. Only one slot shaped (2 X 70 cm) outlet is located in the front seat leg area.

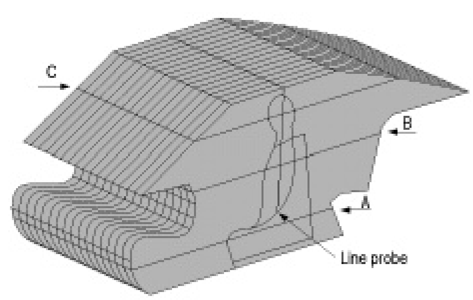


Figure 6. Schematic of the representative cross-sections

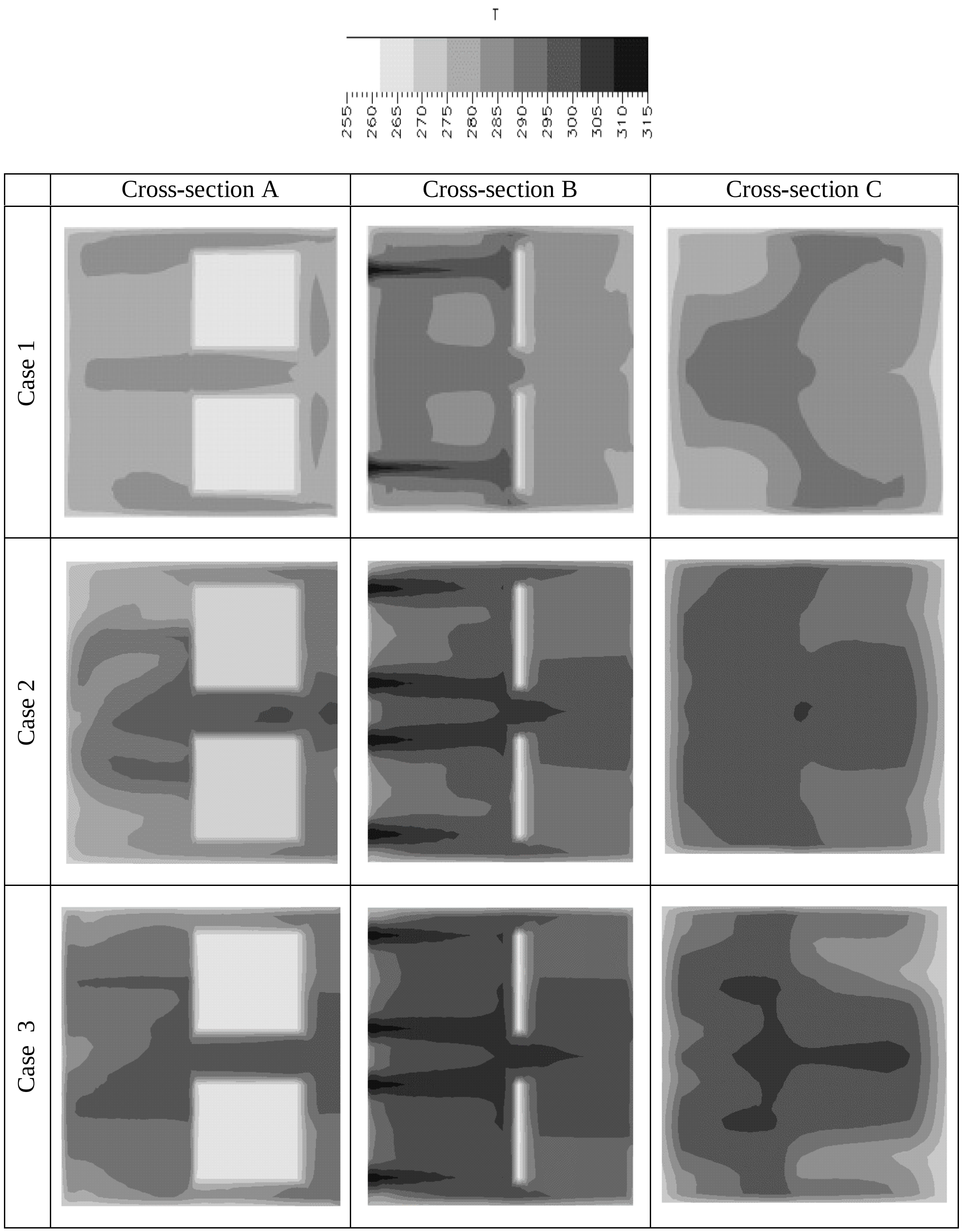


Figure 7. Temperature profiles of three cross-sections of three different cases of inlet-outlet configurations

The rest of the boundary conditions are the same as for the last two-dimensional case presented in figure 4, with the exception to assumption of no passengers in the car.

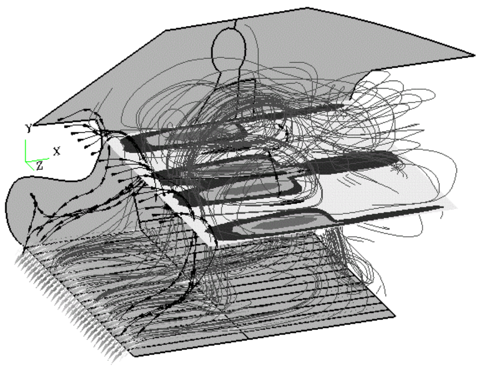


Figure 8. Velocity field inside the cabin (Case 3)

In order to visualise three-dimensional results we make three indicative cross-sections as shown in figure 6. Although we assume that there are no passengers in the car, we place a line probe in the centre of one of the front seats. The direction of this probe coincides with the direction of the passenger's body. The temperature distributions along the probe, to some degree, can substitute for distributions along the passenger's body that, in turn, may determine the level of passenger's comfort.

Simulation results are presented in figures 7, 8 and 9. It can be seen from these figures that HVAC inlet-outlet configuration significantly affects temperature distributions inside a car. Increase of the number of the inlets leads to the increase of the average temperature (figures 7 and 8, case 1 versus case 2) which is not of a surprise. However, what is somewhat less expected is that the right placement of outlets can significantly increase the efficiency of a HVAC system. This is illustrated by case 2 versus case 3 (figures 7 and 9). Although these two cases differ only by the location of outlets, in the third case the average temperature is higher, the temperature distribution is more uniform and the area of "cool" zones is smaller. Three dimensional velocity field is shown in figure 8. Very strong three-dimensionality of the flow is clearly visible. It is manifested through three jets formations (dark area on the flooded cross-section) coupled with recirculations formed in front of the seats. This explains temeparure plots shown in figure 7 (case 3/crossection B). Geometrically continious placement of the exhaust (in z-direction) causes a uniform layer of streamlines to form in the lower front section of the cabin and under the front seats. Strong vortices are formed in front of the "passengers" and most of the windshield flow (marked by arrows) is uniformly directed towards the exhaust.

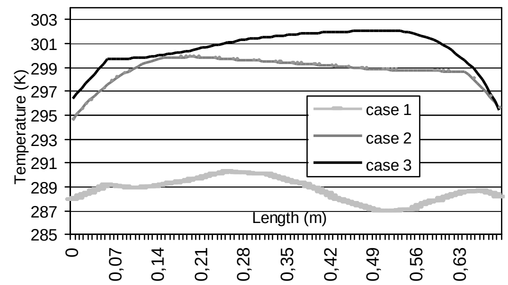


Figure 9. Temperature profiles along the line probe

CONCLUSIONS

In this paper, we have presented results of simulations of a car HVAC system. We have addressed the main parameters that affect the system. The results indicate that both external parameters, such as the car velocity and outside temperature, as well as internal parameters, such as the air velocities and temperatures at the inlets and inlet-outlet configuration play an important role in determining car HVAC system efficiency. The results also indicate that some of negative effects, such as development of zones of low air circulation can be significantly reduced by improving inlet/outlet configuration. However, further study is necessary to account for all parameters that affect car HVAC system performance. Research is under way to address these parameters further.

REFERENCES

[1] D.P. Wyon, S. Larsson, B. Forsgen, I. Lundgren Standard Procedures for Accessing Vehicle Climate with Thermal Manikin, SAE TP 890049, 1989, 1-11

[2] B. Taxix-Reischl, S.Morgenster, T. Mersch, F. Brotz, SAE TP 01-1699 Progress in the Optimized Application of Simulation Tools in Vehicle Air Conditioning, 2001

[3] CFDRC website article - cfdcanada website link: http://www.cfdrc.com/datab/news/articles/ME_3.htm Melting Windshield Ice

[4] CFD-ACE+ , User Manual, Version 6.2, CFD Research Corporation, 2000