

Assessment of Various Environmental Thermal Loads on Passenger Thermal Comfort

2010-01-1205

Published
04/12/2010

Taeyoung Han
GM

Kuo-Huey Chen
General Motors R&D Ctr

Bahram Khalighi
GM R&D Center

Allen Curran, Joshua Pryor and Mark Hepokoski
ThermoAnalytics Inc.

Copyright © 2010 SAE International

ABSTRACT

Virtual simulation of passenger compartment climatic conditions is becoming increasingly important as a complement to the wind tunnel and field testing to achieve improved thermal comfort while reducing the vehicle development time and cost. The vehicle cabin is subjected to various thermal environments. At the same time many of the design parameters are dependent on each other and the relationship among them is quite complex. Therefore, an experimental parametric study is very time consuming. The present 3-D RadTherm analysis coupled with the 3-D CFD flow field analysis takes into account the geometrical configuration of the passenger compartment which includes glazing surfaces and pertinent physical and thermal properties of the enclosure with particular emphasis on the glass properties. Virtual Thermal Comfort Engineering (VTCE) is a process that takes into account the cabin thermal environment coupled with a human physiology model. This model uses multiple body segments and each segment is modeled as several body layers (core, muscle, fat, and skin tissues) and a clothing layer. The UC Berkeley comfort model has the ability to predict local and overall thermal comfort levels of an occupant based on local skin and core temperatures and their time rate of changes. The combination of the present thermal analysis and the thermal comfort evaluations based on the human physiology model will shed light on the climate control strategies as they relate to the

occupant thermal comfort for an “Energy Efficient HVAC System”.

INTRODUCTION

The passenger thermal comfort has been an important subject for the automotive industries. However, the tendency to use more glass in vehicles (styling), tightening fuel-economy constraints, and the changes to the environmentally friendly refrigerants have hampered achieving the occupant thermal comfort. As the automotive industries try to reduce the HVAC power consumption due to the tightening fuel economy, it has become necessary to develop new tools that can predict the impact of various design selections on the passenger thermal comfort early in the design process [1,2,3,4,5]. Thermal comfort is the ultimate goal of the HVAC systems for vehicles. However, assessment of the thermal comfort in a vehicle is very complex due to highly asymmetric thermal environments associated with highly non-uniform air velocity and temperature distributions, localized solar flux, and radiation heat load from the windshield and instrument panel. A passenger compartment thermal analysis tool coupled with the thermal comfort predictions can guide design directions during the early stage of vehicle development process.

During the past two years, the University of California at Berkeley has collaborated with GM R&D Center and TAI

(ThermoAnalytics Incorporated) to develop the capability of predicting passenger thermal comfort to support vehicle climate control systems. At the core of this Virtual Thermal Comfort Engineering (VTCE) tool is the human physiology model coupled with the thermal sensation and comfort correlations developed by UC Berkeley based on the large number of human subject tests [6,7,8]. The human physiology model uses multiple body segments and each segment is modeled as either four or five tissue layers (typically, core and/or bone, muscle, fat, and skin tissues) and a clothing layer. The UC Berkeley comfort model has the ability to predict local and overall thermal comfort levels of an occupant based on the local skin and core temperatures and their time rate of changes. The details of the UC Berkeley thermal sensation and the thermal comfort models can be found in [6,7,8,9,10,11].

The VTCE tool can simulate the occupant skin and core temperatures under various thermal environments (i.e., passenger compartment which is transient and highly asymmetrical thermal environment). It includes solar radiation, mean radiant temperature, local air velocity, air temperature, and humidity. Three-dimensional flow field has been simulated with Fluent CFD code to predict the air velocities around an occupant during soak and cool-down processes. Validation of the three-dimensional numerical simulations has been previously performed for a simplified passenger compartment [12, 13]. In the present approach, the occupant skin and core temperatures were predicted by RadTherm [14] using the air velocities around the occupant obtained by Fluent [15] CFD simulations. The convective heat transfer coefficients around the occupant were estimated based on the empirical correlations in RadTherm by mapping the local air velocities around the occupant body segments. The human body thermal regulation is achieved mainly by regulating blood flow (vasoconstriction and vasodilatation) in order to control skin temperature and increase or decrease the heat loss to the environment. The human physiology model predicts body skin and core temperatures around an occupant by taking into account the human thermal regulatory system combined with the countercurrent heat exchange blood flow model between the arteries and the veins.

The primary objective of the present study is to understand the sensitivity of the various vehicle cabin thermal environments affecting the occupant thermal comfort, including the discharge airflows, glass properties, vehicle body insulation, and the vehicle interior thermal mass for a compact size vehicle. Many of these parameters are interdependent and the relationship among them is complex. The overall performance target for the “Energy Efficient HVAC System” is to minimize the HVAC power consumption without sacrificing the occupant thermal comfort. The desired performance target could not be achieved with only an efficient climate control system. Thus,

to achieve the desired performance, the environmental thermal load and the interior vehicle mass must be reduced.

VIRTUAL THERMAL COMFORT ENGINEERING

In the vehicle environment, many of the design parameters are dependent on each other and the relationships between them are very complex. Therefore, an experimental parametric study is nearly an impossible task. However, with the VTCE tools, the parameters under investigation can be analyzed very easily so that it is possible to vary only one parameter without influencing others. The flow and thermal analysis of a passenger compartment involves not only the geometric complexity but also includes strong interactions between the airflow and three modes of heat transfer, namely, heat conduction, convection, and thermal radiation. Solar radiation introduces heat load to the passenger compartment by transmission through the windshield, backlight and the side windows. With the commercial code RadTherm, we can compute the solar load and the temperature fields in the passenger compartment by mapping the flow field in the compartment from Fluent CFD code. There are seven steps in the VTCE analysis for the “soak and cool-down simulation”:

1. Generation of a the passenger compartment geometry model and meshing
2. Solar load and radiation view factor calculations
3. Determination of the HVAC system transient performance
4. Passenger compartment CFD air flow analysis including an occupant
5. Passenger compartment thermal analysis including an occupant
6. Prediction of occupant skin and core temperatures
7. Prediction of thermal sensation and thermal comfort from the skin and core temperatures

VTCE steps are described in the following sections.

1. Passenger Compartment Geometry Model

The geometry of the passenger compartment was obtained from a UG model, as shown in [Figure 1](#). The geometry model has the key design parameters, such as the A/C outlet location and size, windshield angle, body vent locations, and many other parameters which influence the performance of the HVAC system. Due to the readily available surface geometry from the UG solid model, the interior surface mesh and the volume mesh of the passenger compartment can be created with the ANSA software [16].

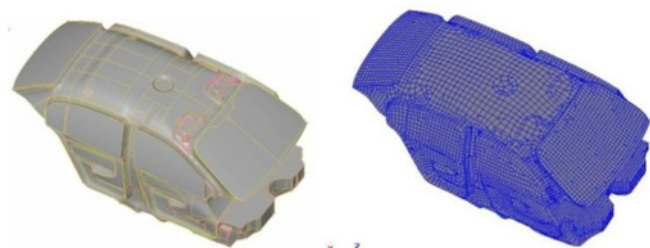


Figure 1. Surface geometry and RadTherm surface mesh representation of the passenger compartment.

2. Solar Load Simulation and Radiation Heat Transfer

The solar load on the vehicle compartment depends on the glass properties and the solar incidence angle. The absorption, transmission, and the reflection of the glass vary depending on the incident angle of the sun. The typical solar intensity varies depending on the time of the day, date, location, and the vehicle orientation. The solar load module in RadTherm keeps track of the reflection from the glass, absorption by the glass, and the transmission into the cabin. The solar flux absorbed by the glass of the passenger compartment is shown in [Figure 2](#). In the present study, the solar load on the exterior body surface was omitted although RadTherm can provide the solar load on the exterior surfaces. The solar flux transmitted into the passenger compartment on the interior surfaces is shown in [Figure 3](#). In the present study, the solar intensity was specified based on the time, date, location and vehicle orientation as summarized below.

Date and time: June 21, from 1:00PM to 2:00PM

Location: longitude= -112, latitude= 33.5 (Phoenix, Arizona)

Vehicle orientation: facing south

The total solar intensity varies with the time of the day. The total solar intensity of 987 W/m^2 was specified at 1:00PM and 961 W/m^2 was specified at 2:00PM. The diffuse solar radiation was accounted for about 117 W/m^2 at 1:00PM and 114 W/m^2 at 2:00PM.

The “surface to surface” radiation heat transfer between the interior surfaces can be obtained by using the view factor calculations within RadTherm. In the present study, the multi-bounce radiosity model in the RadTherm was utilized to handle the radiation heat transfer between the interior surfaces.

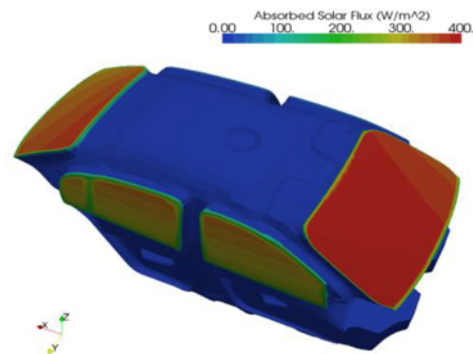


Figure 2. Prediction of solar flux (W/m^2) absorbed by the glasses of the passenger compartment at 1:00PM.

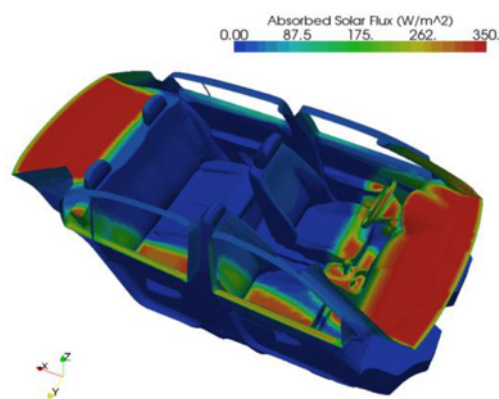


Figure 3. Prediction of solar flux (W/m^2) on the interior surface of the passenger compartment at 1:00PM.

3. HVAC System Transient Performances

The system airflow rate at the A/C and heater outlets provide the boundary conditions for the CFD analysis of the passenger compartment. The system airflow rate and the discharge air temperature for the cool-down and the warm-up analysis can be specified from the A/C system simulation code [17]. In the present study, we used the measured average discharge temperature from all four AC outlets of the passenger compartment, as shown in [Figure 4](#). These temperature profiles were specified in the RadTherm package as a transient temperature boundary condition.

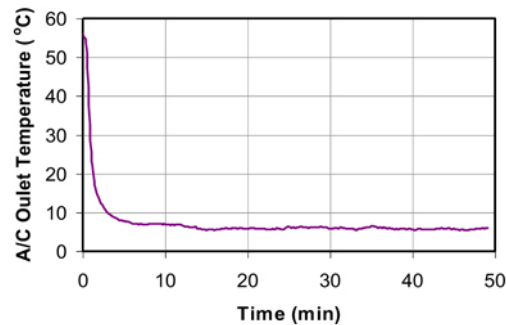


Figure 4. The measured A/C outlet discharge temperature variation with time for “soak and cool-down simulation”.

4. Passenger Compartment CFD Air Flow Analysis

The turbulent nature of the flow is modeled by the realizable k- ϵ turbulence model with the standard wall function [18] including the buoyancy effects. The interior air volume was subdivided into tetrahedral elements using ANSA [16]. About 2,000,000 tetrahedral elements were required to capture the geometric and the flow details of the passenger compartment. The initial air velocity distributions in the passenger compartment were provided from a hot soak (A/C off with solar load) analysis. At the four A/C outlets, the discharge air mass flow rate of 0.136 kg/s was specified throughout the cool-down simulation. At the two exit- air- vents on the rear package shelf, zero temperature gradient condition and constant pressure boundary conditions were specified. At the interior solid surfaces, no-slip boundary conditions were imposed.

5. Passenger Compartment Thermal Analysis

Various thermal boundary conditions can be used for the simulation of the heat transfer through the interior walls. In the present study, the shell element heat conduction heat transfer approach in RadTherm was used by simulating the heat convection and the heat conduction with a finite thickness shell conduction elements. The interior air volume was subdivided into 10 large control volumes using RadTherm. The bulk air flow distribution at 10 control volumes were mapped from the detailed Fluent CFD air flow simulations. The transient cool-down analysis requires initial conditions such as the initial air temperature distributions and the velocity distributions in the passenger compartment as well as boundary conditions. These initial air temperature and velocity distributions in the passenger compartment were provided from the previous work [19] for the hot soak (A/C off with solar load) analyses with Fluent CFD code including the buoyancy effects. At the four A/C outlets, the measured discharge air mass flow rate of 0.136 kg/s and measured discharge air temperatures (as shown in Figure 4) were specified throughout the cool-down simulation. The discharge air temperature between data points was linearly

interpolated. At the exterior solid surfaces, convective heat transfer was used to account for any heat transfer to the exterior environment. In the present study, the outside ambient temperature was specified as 37.5 °C with an estimated convective heat transfer coefficient (between 5 -10 W/m²/K).

6. Prediction of occupant skin and core temperatures

The human physiology model in RadTherm can simulate an arbitrary number of body segments. Each of these segments consists of four or five tissue layers (typically, core and/or bone, muscle, fat, and skin tissues) and a clothing layer. A blood pool node and a series of conductors provide for convective heat transfer between arterial blood and the tissue nodes as well as for the countercurrent heat exchange between the arteries and the veins. The human body thermal regulation is mainly achieved by regulating the blood flow, so a realistic blood flow model is important for any dynamic model of the human thermal comfort. The occupant body uses vasoconstriction and vasodilatation to regulate blood distribution in order to control the skin temperature through an increase or decrease of heat loss to the environment. Veins and arteries are paired, even down to very small vessels, and veins carry heat from the arteries back to the core. The details of this human physiology model are described in [20,21,22,23]. The model is able to predict both core and extremity skin temperatures with a reasonable accuracy under a range of environmental conditions. Detailed validations for transient conditions can be found in [24, 25].

The current model includes clothing nodes to model the heat capacitance and the resistance to the flow of both heat and moisture due to the clothing. Heat capacity of the clothing is important when considering the transient effects [26]. Moisture resistance is important to correctly model the evaporative heat loss from the body through clothing. The human physiology varies significantly among individuals, and these differences can affect the perceptions of thermal comfort; e.g., higher metabolic rate or increased body fat can cause people to feel warmer. The present model which is a standard 50 percentile physiology model was used for the human physiology with a metabolic rate of 2 MET (1 MET= 1 kcal/kg/hr). Clothing level varies based on the body segment - short sleeve shirt, trousers, socks, shoes, etc. for summer clothing during cool-down simulations.

In almost any environment, the occupant body is in contact with solid surfaces and loses or gains heat via heat conduction. In the passenger compartment, the seat contacts a considerable fraction of the body and must be considered to accurately model the occupant. The current model includes a contact surface for each body segment. The thermal properties of the contact surface are used to simulate its surface temperature. Each body segment includes the