

Numerical Investigation of Buoyancy-Driven Flow in a Simplified Underhood with Open Enclosure

Kuo-Huey Chen
General Motors Company

James Johnson
General Motors Company (retired)

Parviz Merati and Charles Davis
Western Michigan Univ.

ABSTRACT

Numerical results are presented for simulating buoyancy driven flow in a simplified full-scale underhood with open enclosure in automobile. The flow condition is set up in such a way that it mimics the underhood soak condition, when the vehicle is parked in a windbreak with power shut-down after enduring high thermal loads due to performing a sequence of operating conditions, such as highway driving and trailer-grade loads in a hot ambient environment. The experimental underhood geometry, although simplified, consists of the essential components in a typical automobile underhood undergoing the buoyancy-driven flow condition. It includes an open enclosure which has openings to the surrounding environment from the ground and through the top hood gap, an engine block and two exhaust cylinders mounted along the sides of the engine block. The calculated temperature and velocity were compared with the measured data at different locations near and away from the hot exhaust plumes. The numerical predictions reveal a complex flow structure under the buoyancy condition, and the results show very favorable comparisons with the experimental velocity and temperature measurements.

CITATION: Chen, K., Johnson, J., Merati, P. and Davis, C., "Numerical Investigation of Buoyancy-Driven Flow in a Simplified Underhood with Open Enclosure," *SAE Int. J. Passeng. Cars - Mech. Syst.* 6(2):2013, doi:10.4271/2013-01-0842.

1. INTRODUCTION

Buoyancy convection heat transfer plays a dominant role in a vehicle underhood environment during "soaking". The underhood thermal soaking, or "soak", condition occurs after the vehicle endures high thermal loads due to performing a sequence of operating conditions such as highway driving and trailer-grade loads in a hot ambient environment (>38 degrees C). The vehicle is then parked in a windbreak, and the power is shut down. Due to the absence of any underhood airflow from the fans or the ambient surroundings, the underhood begins a thermal process that is dominated by buoyancy-driven flow. During the soaking, the temperature can rise even higher in a tightly packed underhood environment.

To numerically simulate the underhood soak condition in a full-size vehicle, a significant amount of computer resources and computational techniques are required. The soak process is inherently transient and involves a small scale

flow motion (usually less than 1 m/s) in a very complex 3D underhood layout. Before this thermal process can be confidently analyzed in a production analysis environment, it is necessary to evaluate the numerical approaches step by step using an idealized geometry that captures the essential features of the flow physics but avoids undue complexity.

Literature survey shows several numerical and experimental studies for underhood buoyancy driven flows in the automobile industry [1,2,3,4,5,6,7,8,9,10]. The work published by Franchetta, et al. [1] concentrated on experimental and computational investigation of a half-scale underhood compartment using PIV, thermocouples, and commercial software VECTIS. Study [1] was carried out for the transient natural convection condition with openings on the engine compartment model. Franchetta, et al. [2] used the data obtained in [1] to validate a developed procedure within VECTIS predicting the "soak" condition with substantially reduced CPU time compared with standard fully transient

CFD codes. In Ref. [3,4,5,6,7,8,9], a simplified underhood was used to conduct a series of experimental and numerical studies to characterize the flow and temperature fields under buoyancy-driven flow condition. The experimental underhood geometry, although simplified, consists of the essential components in a typical automobile underhood undergoing the buoyancy-driven flow condition. It includes an enclosure, an engine block and two exhaust cylinders mounted along the sides of the engine block. Under a closed-enclosure condition, Chen et al. [3] conducted a detailed numerical study for the flow configuration with quarter-scale size experimental model used in [4]. Several numerical issues, including mesh type, mesh size, spatial and temporal accuracy, were discussed [3]. For the quarter-scale case, both experimental measurement and numerical simulation exhibit steady state flow pattern. Furthermore, experimental and numerical studies on a full-scale size underhood buoyancy flow were reported in [5,6,7,8]. In this study, the full-scale model with closed enclosure configuration produces quite different flow and temperature fields than does the quarter-scale with a similar temperature boundary condition. Among others, the most significant difference is that both experimental and numerical predictions for the full-scale case reveal a complex, transient flow structure which does not exist for the quarter-scale case. Detailed numerical analysis was reported in [6] and [8] and the transient behaviors of the flow field compared very closely with the experimental data. A most recent experimental investigation was reported [9] on a full-scale simplified underhood with open enclosure to the surrounding environment. The openings used in [9] are used to mimic the ground air entrainment effect and to represent the air gap between the hood and the fender in the vehicle.

In the current paper, a three-dimensional, transient simulation technique was applied to validate the buoyancy-driven flow and temperature fields for the full-scale, simplified underhood with open enclosure conducted experimentally in [9]. This paper is organized as follows. A brief overview of the experimental setup and measurement is presented first. Following that, the numerical study using ANSYS Fluent [10] and the comparisons with measurement data are discussed.

2. EXPERIMENTAL SETUP

The details of the experimental work were well documented in [9]. In the current paper, the experimental setup and data will be described briefly so that the numerical study can be easily explained. The experimental configuration is shown in Fig. 1 which depicts an outer enclosure with openings, an engine block and two exhaust cylinders mounted along the sides of the engine block. The size of the outer enclosure in the simplified underhood shown in Fig. 1 is roughly $1\text{ m} \times 1\text{ m} \times 1\text{ m}$ and it has a slanted hood and sloped dash which mimic the underhood compartment in a production vehicle. The two exhaust cylinders are identical in length and size, and they are 584.2 mm long and 45.7 mm

in diameter. A brief sketch with dimensions for the experimental setup is shown in Fig. 2.

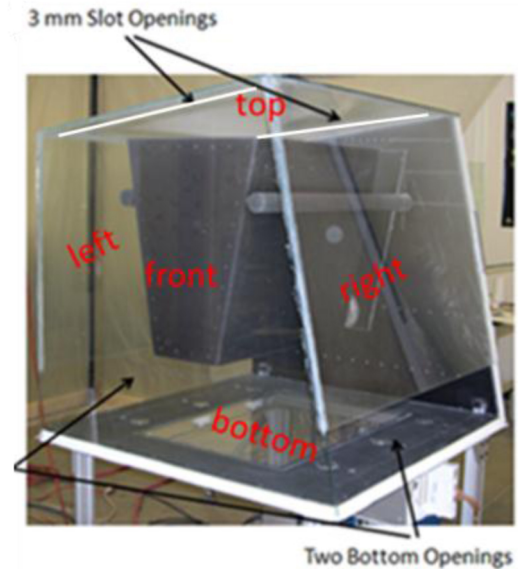


Figure 1. Experimental setup

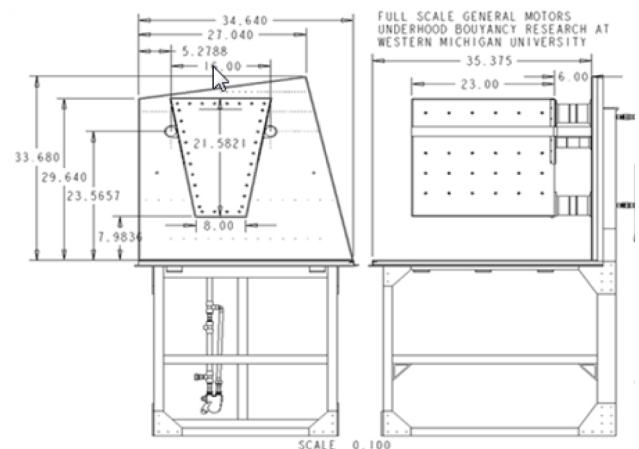


Figure 2. Overall dimensions of the experimental underhood (units in inches)

Wall Boundary Temperature Conditions

There are two types of boundary surfaces in which temperature distributions need to be experimentally controlled. These are: exhaust cylinders and engine block. The surface temperature of each exhaust cylinders was kept at $600\text{ }^{\circ}\text{C}$ by inserting a 3500 W cartridge heater into the core of the cylinder. The surface temperature of the engine block was kept at a nominal temperature of $100\text{ }^{\circ}\text{C}$ by a specially designed water spray circulating system inside the engine block. A total of 128 thermocouples were buried under the skin of the engine block to monitor and record the actual temperature distribution on the engine block surfaces. As can be seen later, some surface areas, due to the proximity to the

two exhaust cylinders, are inevitably hotter than 100 °C. The surface temperatures of the outer enclosure were measured by using thermocouples directly placed on the enclosure surfaces to record the final equilibrium temperatures.

Air Temperature and PIV Flow Measurements

Air temperature

The air temperatures were measured by thermocouples at 104 locations inside the engine compartment. The 104 thermocouple locations are shown in Fig. 3 on the x mid-plane (at x = 444 mm). The x-y-z coordinate directions are shown in Fig. 4. There are five horizontal rows with the 1st row near the top and the 5th row near the bottom. Two vertical columns pass through the center of the left and right exhaust cylinders, respectively on the same x mid-plane. The locations of the five horizontal rows and two vertical columns are indicated in Fig. 3.

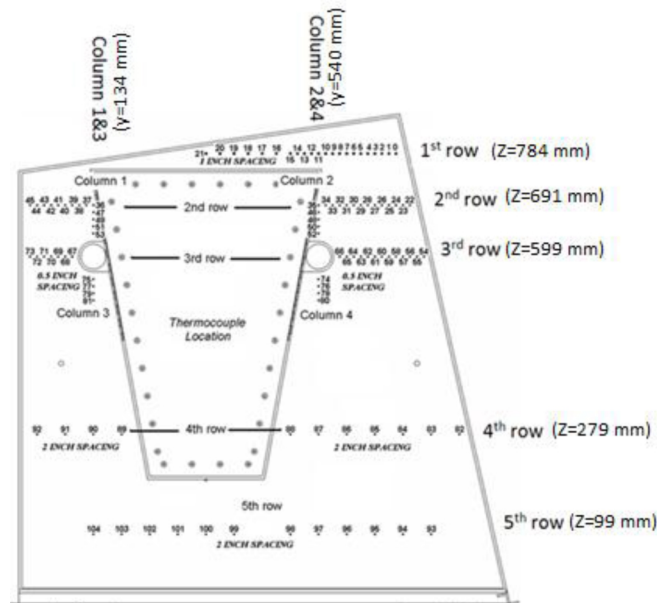


Figure 3. Locations of the air temperature measurements

PIV flow measurement

The air velocity was measured by a conventional 2D PIV (Particle Image Velocimetry) system. The PIV setup can be seen in Fig. 4. There are five PIV test planes in the x-direction that correspond to x mid-plane, front quarter plane (x=571 mm) and back quarter plane (x=317 mm). The other two planes shown in Fig. 4 are at x = 70 mm and 830 mm, respectively. There are also other four PIV planes not shown in Fig. 4 in the y and z directions. See Ref. [9] for details.

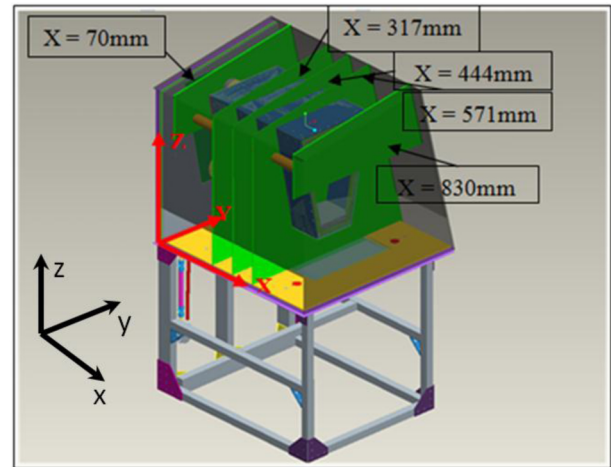


Figure 4. Experimental setup of PIV

Surface Temperatures for Numerical Boundary Conditions

The temperature maps on all the surfaces are recorded. These include exhaust cylinders, engine block, outer enclosures and the three pipes which provide the heating and cooling fluids and the thermocouple wiring in and out of the engine block. Figure 5 shows the inner surface temperature distribution of the outer enclosure. The temperature distribution is quite non-uniform with strong influence from the exhaust cylinders on both sides. Figure 6 shows the temperature distribution for the surfaces of the engine block. Table 1 lists the temperature statistics of the engine block. Again, the surface temperatures on the engine block are much higher than the designed 100 °C due to the proximity to the hot exhaust cylinders. The surface temperature of both exhaust pipes is set to be 600 °C. Table 2 lists the temperature statistics of the outer and inside surfaces of the enclosure. The experimental measurements are used as input boundary conditions for the simulations.

Table 1. Temperature statistics for each surface of the engine block

Engine Block Surface Temperatures C°						
	Front	Back	Right	Left	Top	Bottom
Mean	90.1	99.4	107.3	105.7	100.3	99.0
Std Dev	1.6	1.6	7.7	4.5	0.2	0.4
Max	92.1	100.5	127.7	117.5	100.6	99.4
Min	86.1	94.4	98.8	99.9	100.1	98.2
Range	6.0	6.1	28.9	17.7	0.5	1.2

Table 2. Surface temperature statistics for each surface of the outside and inside of the enclosure

	Glass Outer Surface Temperatures °C			
	Front	Right	Left	Top
Mean	53.3	80.0	90.3	88.7
Std Dev	11.6	24.7	33.4	26.1
Max	76.6	123.8	146.2	148.1
Min	30.7	38.7	37.2	61.6
Range	45.9	85.1	109.0	86.5

	Enclosure Inner Surface Temperatures °C	
	Back	Bottom
Mean	56.4	48.9
Std Dev	8.0	6.9
Max	78.7	64.7
Min	43.5	39.4
Range	35.2	25.3

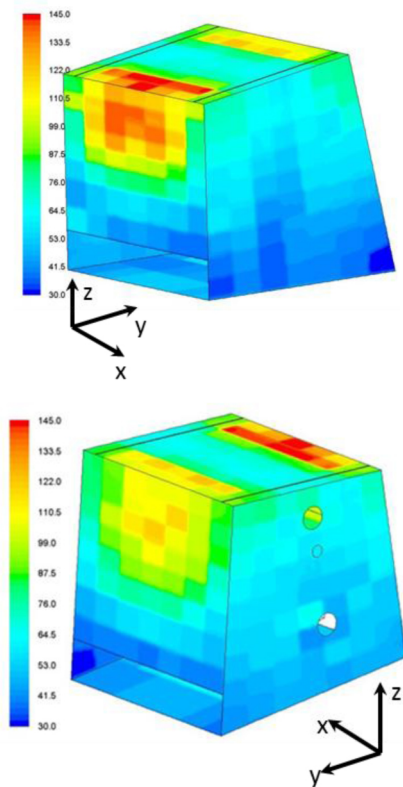


Figure 5. Temperature maps for the inside surfaces of the enclosure.

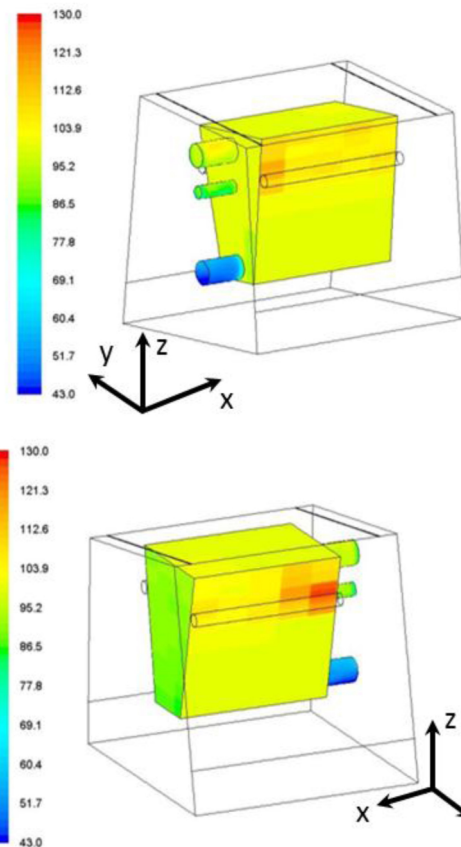


Figure 6. Temperature maps for the engine block.

3. NUMERICAL RESULTS

Numerical Setup

Numerical simulations were carried out with Fluent version 14 [10]. From the previous closed full enclosure study [6], it was concluded that the 2nd order accuracy in both space and time is best suitable for the present simulation. It also demonstrates that a well prepared tetrahedral mesh with or without the prismatic mesh can achieve a comparable accuracy. The experiences gained from the detailed numerical study [6] lead to the following mesh size arrangement. There are three mesh densities, labeled “coarse”, “medium” and “fine” meshes, used for the numerical study. On the engine block and the exhaust cylinders, finer surface meshes were used. The mesh sizes are 1 mm, 2 mm and 5 mm, respectively for three mesh densities. Prismatic meshes were used in the current study with four layers extruding from the engine block and the exhaust cylinder surfaces. The first layer was 0.25 mm from the wall surfaces. In general, coarser surface meshes were used on the outer enclosure surfaces since they are farther away from the high temperature gradient region. The mesh size ranges from 3 to 10 mm depending on the location. The rest of the domain was filled with tetrahedral meshes. As a result, three cases with total mesh count of 2 million (coarse), 7.5 million (medium) and