Visualization of Thermal Flows in an Automotive Cabin with Volume Rendering Method

Kenji Ono[1], Hideki Matsumoto[2], and Ryutaro Himeno[3]

[1] Vehicle Research Laboratory, Nissan Research Center,
1, Natsushima-cho, Yokosuka-shi, 237-8523, Japan
kj-ono@mail.nissan.co.jp
[2] NEC Informatec Systems,
3-2-1, Sakato, Takatsu-ku, Kawasaki-shi, 213-0012, Japan
matumoto@ssd.nis.nec.co.jp
[3] The Institute of Physical and Chemical Research,
2-1, Hirosawa, Wako-shi, 351-0198, Japan
himeno@postman.riken.go.jp

Abstract. A predictive system of thermal flow with quick turnaround time in a passenger compartment has been developed. An efficient method based on the Cartesian mesh system was used to reduce the period of analysis. The computed temperature in an automotive cabin was visualized by volume rendering techniques using an *RVSLIB* software library developed by NEC. Consecutive images of the flow were converted into MPEG1 movies, which gave us an overall understanding of the flow. The visualization results indicate that the present system is capable to sufficiently predict the thermal environment in a vehicle cabin at early stage of vehicle development.

1 Introduction

Thermal environment in a passenger compartment is an important issue in the interior design for the comfort of passengers, the reduction of stress of a driver, and the visibility. The thermal environment in a cabin is strongly influenced by sunlight, radiation of heat and ventilating flow from HVAC unit. Many researchers have studied the thermal flows in passenger compartment [1–6]. Generally, the exterior and the interior design that affect the thermal environment are determined at the early stage of vehicle design and therefore, it is important to predict the temperature distribution at this stage. Also, from the viewpoints of both the reduction of the development cost and the improvement of the efficiency, application of simulation at the early stage is of great significance. On the other hand, the vehicle shape at the early stage of vehicle development is neither accurate nor detailed. Moreover, the design may be changed every day. Thus, application of the simulation at the early stage can even dominate the direction of the vehicle development instead of trial and errors. The essential issues required for the analysis are the short turnaround time and the accuracy of the computed results. In this respect, it is more desirable of qualitative judgment in short period other than high accuracy for the prediction of the vehicle performance. For actual flow applications, there is no choice of commercial packages available because the mesh generation is very much time-consuming due to the geometric complexity. For instance, in a recently published paper that reported the thermal flow prediction with realistic configuration [4], the mesh system was generated for the whole shape, from an air duct system to a passenger compartment with four occupants, resulting in about 5 million hexahedra meshes. It was said that it had taken approximately 3 months to get the whole mesh systems completed. In order to overcome the problem of mesh generation, the authors have developed a flow solver and a mesh generation system, based on the Cartesian mesh method, which is capable to perform automatic operation [7]. This system was applied to the prediction of the thermal flow in a vehicle cabin. Concerning the visualization, there are many techniques such as streamline, contour line and so forth. In order to express the temperature distribution in a three-dimension manner, the spatial distribution is represented by combining several cross-sections, which depend on conjecture. Iso-surface technique may be used, however the information of the depth direction disappears behind the closest surface. A volume rendering method can represent the depth information as like cloud. And this method has advantage to provide intuitive image of the flow structure. Although the volume rendering is a sophisticated technique, it is hardly used for the actual design because of its complicated operation and extra computation time. Fortunately, we have a highly vectorized volume rendering library [8]. In this paper we report some preliminary test cases of the volume rendering visualization and furthermore discuss its computational performance in applying the rapid simulation to evaluation of the temperature distribution in a car cabin.

2 Voxel Modeling Method

Examining the thermal flow in a cabin need to consider the location and the direction of the ventilating opening as well as the distribution of the airflow rate for each opening. Additionally, it is important to study the temperature distribution around both driver and other occupants. The complicated shape should be treated in a realistic manner. Unfortunately, we still have no method that can generate the mesh systems for the complex shape efficiently. Generally, it needs tremendous efforts and hence is very much time-consuming. In order to largely shorten the period of analysis, a voxel modeling method is hereby employed to reduce the time in mesh generation. The voxel method approximates the shapes of objects in a step form and has only information on whether the cell is included in the objects or not [9]. The shape of the object is projected onto the Cartesian mesh through a simple judgment, i.e., whether the surface data that represent the shape cut the mesh lines or not. Therefore, mesh can be generated very quickly. Another important advantage is stability. The surface data of a vehicle are not clean. For example, some data are not interconnected, representing unclosed surfaces that should be closed. In the Cartesian mesh approach, shapes under the sub-cell are ignored. This feature acts as a filter for stability and leads to robustness, making it easy to generate mesh automatically. The flow around the approximated shape is solved instead of the original shape. On the other hands, this modeling method has disadvantage that the voxel method can never represent the accurate shapes of the objects because of the approximation by means of cubes. However, it is useful to know the outline of the flow fields in the case with little influence of the flow separation. This method enables us to investigate the flow quickly at the first stage of the vehicle design. Fig. 1 shows the geometry of a production car model used in this study. The shape in the cabin is represented by many polygons as shown in Fig 2. Projecting the polygons onto the Cartesian mesh we obtain the voxel model as illustrated in Fig. 3. This voxel model has a mesh size of 10mm and the total number of meshes is about 8.4 million cells ($171 \times 349 \times 141$). In-house software is used for the generation of the voxel model. All processes of this mesh generation procedure are automated because of their simple nature. The measured time of the conversion from 10 million polygons to 8.4 million cells is just 60 seconds on a PC with an Intel Pentium3 600MHz CPU.

3 Computational Method and Visualization Method

There are many factors that influence the thermal environment in the cabin such as sunlight, radiation and airflow from ventilating opening and so forth. In this paper, the thermal flow only from the ventilating opening is considered because it is the only controllable factor and most significant. The effect of the buoyancy caused by the temperature difference is negligible. A three-dimensional unsteady in-compressible viscous flow is assumed in this study. The governing equations in non-dimensional conservation form are the Navier-Stokes equation, the continuity equation and the energy equation like the convection-diffusion equation of the passive scalar, which is derived from the incompressible assumption. These equations are discretized by a finite volume method on a staggered mesh system. A second-order-accurate QUICK scheme is used for the convection terms and other spatial terms are discretized by central differencing manner [9]. An Euler explicit scheme is employed for the time marching method and a fractional step method [10] is used for the coupling procedure between pressure and velocity fields. The outflow boundary condition is imposed on the outer boundary of the computational domain. At wall, no-slip condition, the Neumann condition and an adiabatic condition are used for velocity, pressure and temperature, respectively. At the boundary of the ventilating opening, both velocity and temperature are given. Fortran77 and C that provide the function of the dynamic memory allocation and an efficient file I/O interface write the program. The vector ratio of the code is over 99.6% and the performances are shown as in Table 1. The measured performance is about 1.8GFLOPS on an NEC SX4 and 3.7GFLOPS on an NEC SX5. The memory size is about 666MB(single precision) for the flow and temperature calculation using the voxel model as in Fig. 3 because this program use 26 words per cell. And if we use the *RVSLIB*, additional 1,038MB will be requested in a double precision format. The flow fields were visualized using the *RVSLIB* developed by NEC [8]. The *RVSLIB* works on network environment as the server-client type application, i.e. the server runs on a high-performance computer and the clients are operated on the user PC. Functions of streamline, pressure distribution, contour line, and volume rendering are provided by a software library on the server. For users what to do is only to call the library interface from the user program. This library visualizes the flow at the same time and generates an image as a result. This is the reason why this visualization system is suitable for the visualization with large-scale data because the data size is almost independent of the data scale. Further information about the *RVSLIB* is available in the literatures [8, 11]. Firstly, the voxel model is assembled on the local PC using the surface data based on the CAD system, and the resultant file of the voxel model is compressed by GZIP. The compressed file becomes rather small, for example, the size of the voxel model of Fig. 3 is only 217 kB. Then, the compressed files are sent to the remote server machine via the Internet. Secondly, the flow computation and the visualization are performed on the server. After the calculation, the stored time sequential images are converted into an MPEG1 movie file. Then, the movie file is transmitted to the local PC and the flow behavior is observed. For the volume rendering visualization, it is required that the set up of a frame, a color map, an intensity of the permeability of the volume and so forth should be generated in advance. Moreover, the parameter survey must be given in order to represent the characteristics of the flow fields. In this respect, the volume rendering visualization needs twice runs in the case of the first try for the target flow.

4 Results and Discussion

Before we discuss about the temperature field, the computed accuracy should be demonstrated. As the flow considered in this paper is incompressible, the passive scalar is strongly influenced by the velocity distribution. The time-averaged distribution of the flow velocity was compared in a preliminary computation as shown in Fig. 4. The flow was blowing from a duct under the front seat in this calculation. The velocity profile is measured just in front of the rear seat. It was found that the peak of the jet flow is well simulated and the overall agreement is very good. Next, we studied the changes of the flow fields and the temperature fields in the cabin according to the ventilating velocity. The interested parameter is the velocity at four ventilating openings on the instrument panel as shown in Fig. 5. The velocities at the openings were changed for the comparison of the flow field. The time dependent temperature fields were examined in this simulation. The influence of the sunlight and the radiation is not considered but only the influence of the convection from the ventilating openings. The Reynolds number of the computed flow is about 2×10^4 and the Pecret number is about 1.44×10^4. The flow fields are scaled by a reference length L_0=0.07 (m) and a reference velocity U_0=5 (m/s). Two cases in Table 2 were computed until 15 seconds in real time and the results were compared. In the case A, the flow velocities at the center ventilating openings are high while the velocities of the both sides are higher than others in the case B. The initial temperature in the cabin is 45 degrees in Celsius and the airflow tempera-ture of four ventilating openings is assumed 15 degrees. This situation is for summer cool-down case. The computed rendering images are demonstrated from Fig. 6 to 11. The color shows the temperature distribution in those images. The color for the highest temperature, i.e. 45 degrees, is transparent. If the temperature is slightly low, the color becomes red. On the contrary, the color for the lowest temperature, i.e. 15 degrees, is blue. The time evolution of the temperature distribution for case A is shown in Figs. 6 - 8 and case B is shown in Figs. 9 - 11. These snapshots show that the temperature around the left person at the rear seat was quite different from each other. From the movie shown in Fig. 11, it was observed that the flow circulates from outside to inside in the cabin. Finally, the turnaround time of the analysis period is referred. The computation time for the flow calculation is about 36 hours on the SX5 (2CPUs) including the generation of the visualized image. In the present case, the image was written to the file every 30 steps during the flow computation. The ratio of the computation time and the visualization time was about 2:1. The time to make the movie is about 30 minutes on an SGI Onyx2 reality monster 16CPUs but it may use only one CPU for its own program *dmconvert*. The file size of the movie with rather high quality is ranging from 6 to 9 MB, it depends on the rendered image. The transfer time of the movie via Internet is about 10 minutes. Consequently, the total time for this analysis is less than 40 hours because the assemble time of the voxel model is very short. This turn-around time is short enough for the educational purpose although we still have many parameters for the thermal flow computation in the passenger compartment. Further reduction of the computation time may be desired.

5 Concluding Remarks

This paper describes a method of thermal flow computation with a voxel model so as to reduce the turnaround time of the analysis. Volume rendering techniques are employed by using the *RVSLIB* visualization system. The present method was applied to compute thermal flows in a passenger compartment to investigate the thermal environment in the cabin. The results indicate that the present visualization system has capability to reasonably predict the thermal environment in vehicle cabin at early stage of vehicle development where the shortening turnaround time is greatly required. For further improvement of the present system, the following issues may need to be resolved:

- The set-up procedure is still complicated and simpler interface is expected.
- Current version of *RVSLIB* requests large memory space.
- Parallel computation of the volume rendering is attractive feature.

This system will be used in a vehicle design in the near future in our company and be deployed by the end of 2001.

References

1. Hara, J., Fujitani, K., and Kuwahara, K.: Computer Simulation of Passenger Compartment Airflow, SAE paper 881749(1988)
2. Lin, C.H., et al.: An Experimental and Computational Study of Cooling in a Simplified GM-10 Pas senger Compartment, SAE paper 910216(1991)
3. Currle, J.: Numerical Simulation of the Flow in a Passenger Compartment and Evaluation of Thermal Comfort of the Occupants, SAE paper 970529(1997)
4. Currle, J., et al.: Evaluation of the HVAC System of Passenger Cars and Prediction of the Microclimate in the Passenger Compartment by Application of Numerical Flow Analysis, SAE paper 971788(1997)
5. Currle, J and Maue, J.: Numerical Study of the Influence of Air Vent Area and Air Mass Flux of the Thermal Comfort of Car Occupants, SAE paper 2000-01-0980(2000)
6. Aronson, D., et al.: Comparison Between CFD and PIV Measurements in a Passenger Compartment, SAE paper 2000-01-0977(2000)

7. Ono, K., Fujitani, K., and Fujita, H: Applications of CFD Using Voxel Modeling to Vehicle Development, Proc. of the 3rd ASME/JSME Joint Fluids Engineering Conference, FEDSM99-7323 (1999)
8. Muramatsu, K., Matsumoto, H., Takei T. and Doi, S.: A Real-Time Visualization System for Computational Fluid Dynamics on Parallel Computers, Proc. Parallel CFD '98, Elsevier Science(1998)
9. Ono, K., Akabane, K., Shiozawa, H., Fujitani, K.: Prediction of Cooling Flow Rate through the Front Grille Using Flow Analysis with a Multi-Level Mesh System, FISITA World Automotive Congress, F2000H201 (2000)
10. Chorin, A.J.: Numerical Solution of the Navier-Stokes Equations, Math. Comput. Vol. 22 (1968) 745–762
11. Takei, T., Matsumoto, H., Muramatsu, K., Doi, S.: Parallel Vector Performance of Concurrent Visualization System *RVSLIB* on SX-4, the 3rd pacific symposium on flow visualization and image processing, March 18-21, Maui, Hawaii (2001)

Table 1. Measured performance of the solver on high-performance servers.

	Peak Performance	Measured Performance
NEC SX4 (2CPUs)	4 GFLOPS	1.8 GFLOPS
NEC SX5 (2CPUs)	8 GFLOPS	3.7 GFLOPS

Table 2. Specified parameters of the non-dimensional velocity for each computation cases.

Opening location	Case A	Case B
Left	0.8	1.6
Center left	1.4	1.0
Center right	1.4	1.0
Right	0.8	1.6

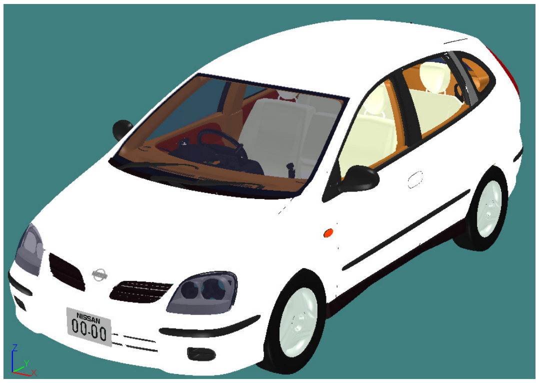

Fig. 1. Geometry of a production car model. This surface data is based on a CAD system and consists of over 15 million polygons.

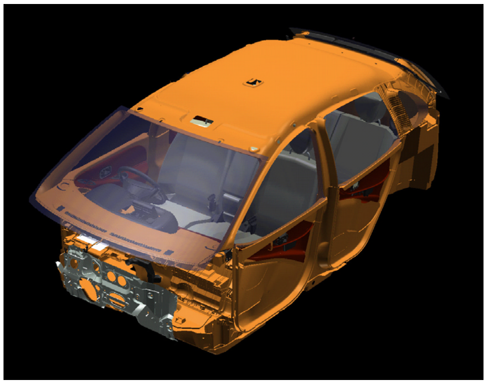

Fig. 2. The extracted surface of the passenger compartment is rendered. The total number of polygon is over 10 million.

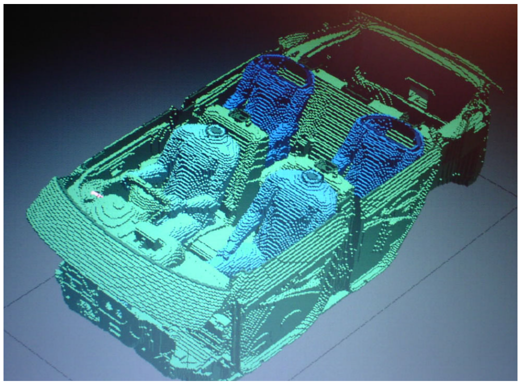

Fig. 3. The voxel model converted from Fig. 2. The total number of voxel is about 8.4 million with 10mm in size. The upper half area is invisible for the display.

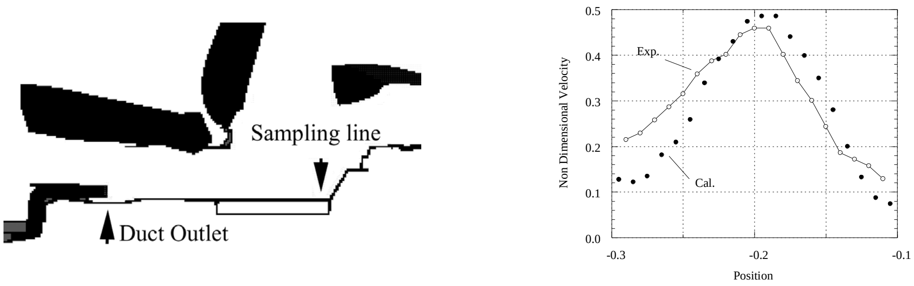

Fig. 4. The left panel shows a cross-section from the left view and measured position. The right panel shows the comparison of velocities. The velocity is normalized by the velocity at the outlet.

Fig. 5. The front instrument panel from the rear view. Two ventilating openings are located at center and other two openings are at both sides near the front pillar. The front seat is removed for convenience.

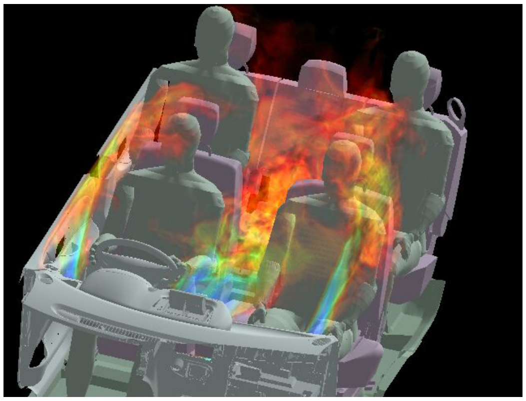

Fig. 6. The temperature distribution of case A after 5 seconds past from the initial state.

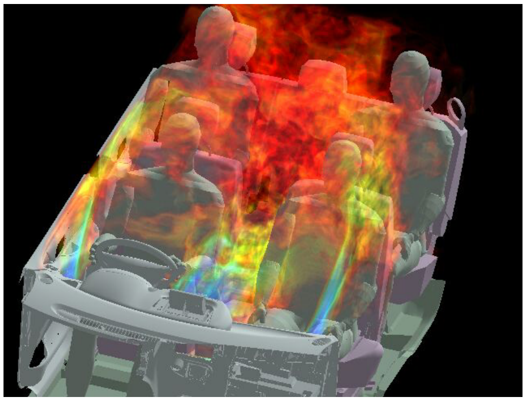

Fig. 7. The temperature distribution of case A after 10 seconds past from the initial state.

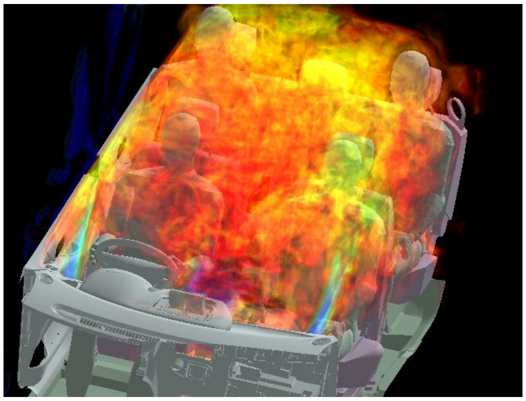

Fig. 8. The temperature distribution of case A after 15 seconds past from the initial state.

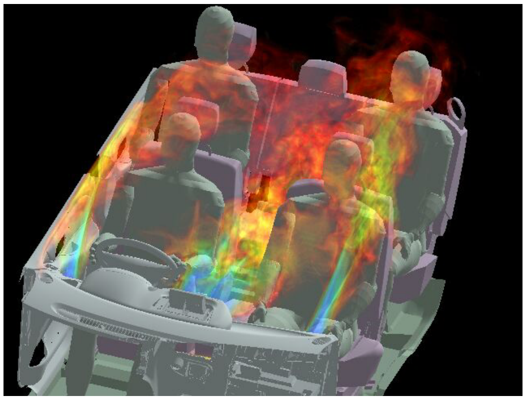

Fig. 9. The temperature distribution of case B after 5 seconds past from the initial state.

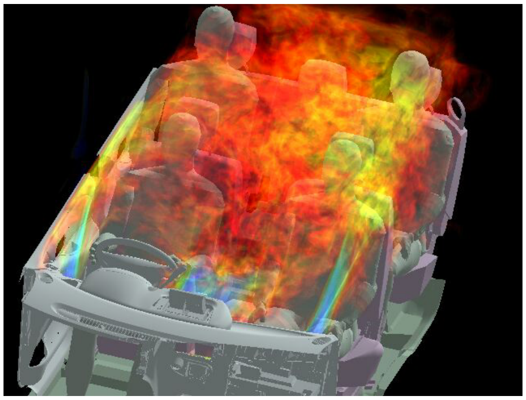

Fig. 10. The temperature distribution of case B after 10 seconds past from the initial state.

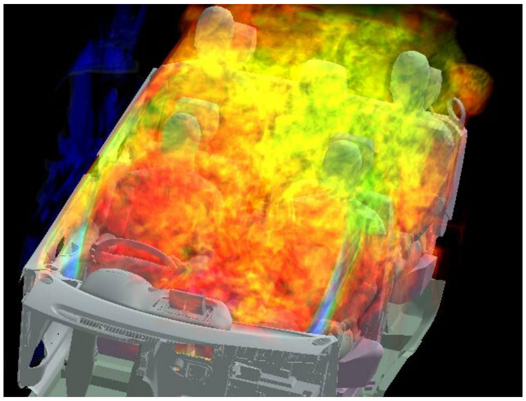

Fig. 11. The temperature distribution of case B after 15 seconds past from the initial state. Click figure, then you will find the thermal flow behavior in the passenger compartment.