 SL, vol.6, no.2, pp.55-64, 2011

Air Quality Modeling in the Automobile

B. Mei[1] **and X. Wang**[2]

Abstract: Automobiles are used every day and air quality in the automobile is directly related to people's health. This term paper focuses on the distribution of automotive interior carbon dioxide on the highway. A typical sedan is chosen as a sample. Carbon-dioxide meter and wind meter are used to get essential data. Then, the data is analyzed by the commercial CFD software — GAMBIT® and FLUENT®. After plotting and comparing the contours of the distribution, how changing the fan speed will change the distribution is got.

Keywords: Air Quality Modeling, Computational Fluid Dynamics, Automobile

1 Introduction

In the US, people spend much time driving automobiles, and they concern more about the health effects of the air quality inside automobiles than ever before. Some studies show that the major pollutants inside an automobile include: particulate matter, volatile organic compounds, carbon monoxide, nitrogen oxides, ozone and so on. All of these pollutants generally come from several major sources: exhaust from other automobiles, general air pollution, auto interior materials, people, and sometimes also from food or pet. These dangerous pollutants can also cause some severe disease even cause the cancer. Nowadays many experiments conclude that air pollution is heavier inside than out. Furthermore, in-car air pollution is becoming one of the great modern threats to human health. Thus, the objective is to analyze the distribution of automotive interior carbon dioxide. This objective is under three conditions. First, the experiment is implemented on the highway. The result is only meaningful under highway conditions. Second, the outside mode of the automobile is on, which means outside air will come through into the automobile. Third, the only variable is fan speed. The focus will be finding the relation between distribution and the wind speed from the vent.

[1] Purdue University Calumet, Hammond, IN, USA
[2] Purdue University Calumet, Hammond, IN, USA

2 CFD Analysis

2.1 Geometry Construction and Data Acquisition

2.1.1 Geometry Construction

2006 Chevrolet Impala is set as a sample. After doing the simplification and measurement, the dimensions of the automobile are shown in Figure 1.

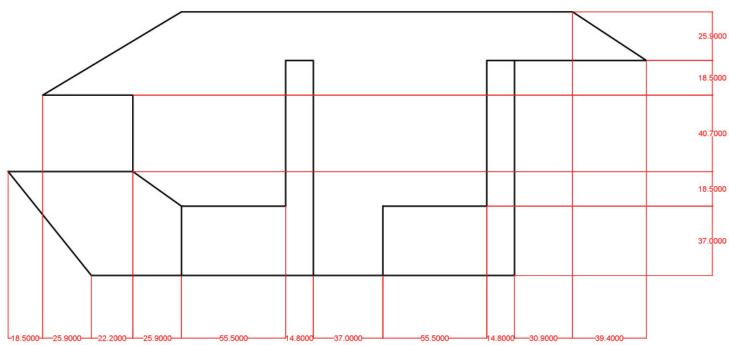

Figure 1: Dimensions of Side View

2.1.2 Data Acquisition

First, the lowest fan speed is 0.5m/s and the highest fan speed is 3m/s. This is got under driving at a speed of 70mi/h on the highway. One fact that should be noted is that although adjustment knob on the instrument panel is set as no wind blowing, wind will still come out from the vent. The reason is the outside air mode is on. Second, carbon dioxide blowing out of the vent is from the outside because of the outside air mode. Thus, the concentration from the outside is measured. The value is 443PPM at the service plaza. Third, the wind speed from the nose is also measured. The average is about 0.3m/s. Fourth, from the literature search, concentration of carbon dioxide from exhalation for human beings is about 4%.

2.2 Mesh Generation

According to the geometry, a model is built in GAMBIT®. Figure 2 shows the model after meshing. Tetrahedral scheme is used. The mesh size is 10. Total mesh number is 27700.

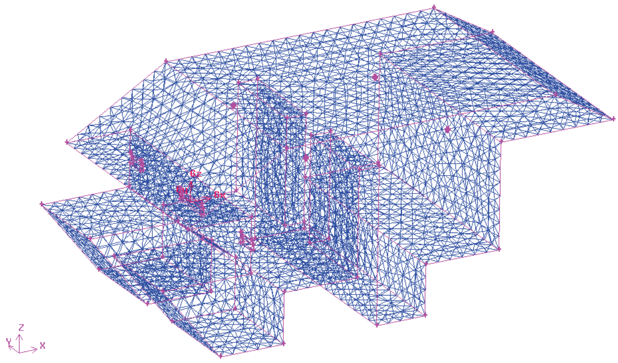

Figure 2: Stereogram With Mesh

2.3 Boundary Condition Setting

Figure 3 shows the instrument panel of the automobile. Because the vent is the only place that connects with the outside, it is divided into two parts. The upper part is set as the velocity inlet and the lower part is set as the pressure outlet. For velocity inlet, the speed changes between 0.5m/s and 3m/s. The concentration of carbon dioxide is constant as 443PPM. The four small cubes shown in Figure are used to simulate the exhalation of breath of four people sitting in the automobile. For each cube, just the front face is set as the velocity inlet. Both speed and the concentration are constant, which are 0.3m/s and 4% respectively. The other faces of the whole volume are set as walls.

2.4 Simulation Results

Six cases are tested in FLUENT®. The difference among them is only the fan speed, the value of which is 0.5m/s, 1m/s, 1.5m/s, 2m/s, 2.5m/s and 3m/s. The models in FLUENT® are viscous — standard k-epsilon and species — species transport. The gravity is turned on. The side view at symmetry plane will be used to do the discussion.

2.5 Mesh Independence

To do the mesh independent study, the mesh size is changed to 3 in GAMBIT®. Figure 4 shows the model after meshing. Under Tetrahedral scheme, the total mesh

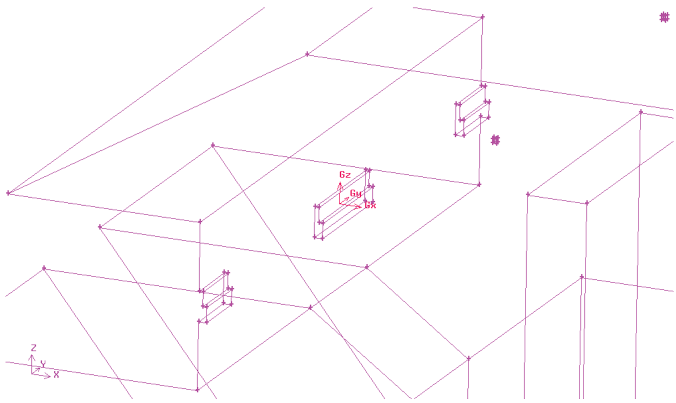

Figure 3: Instrument Panel

number becomes 710675. For comparison, fan speed is set as 3m/s for both cases. Figure 5 shows the concentration of carbon dioxide at the symmetry plane when mesh size is 10. Figure 6 is the situation when mesh size is 3. From both cases, the concentration distributions of carbon dioxide are similar and the scales are the same. Thus, meshing is independent.

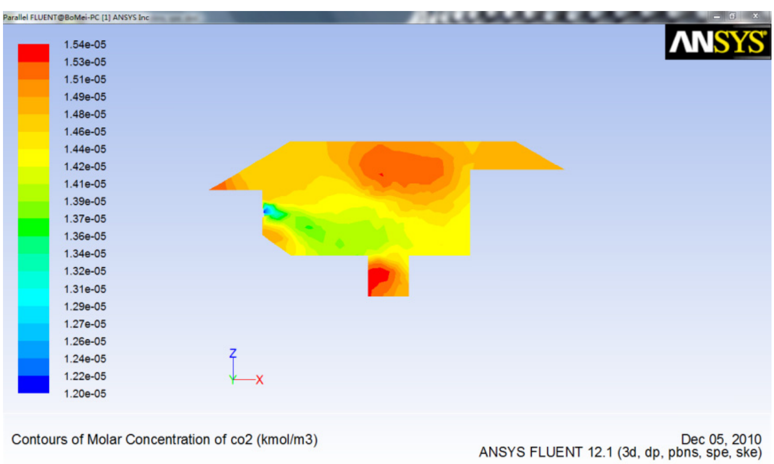

Figure 4: Concentration of Carbon Dioxide (Case 6)

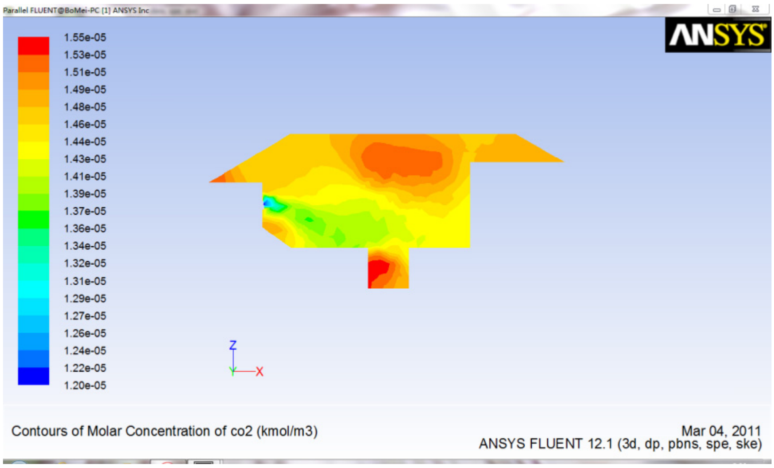

Figure 5: Concentration of Carbon Dioxide (Case 6')

2.5.1 Velocity Vector

Figure 6 and Figure 7 show the velocity vectors for case 1 and case 6. The density of vectors becomes more intensive from case 1 to case 6, which indicates the wind velocity inside the automobile becomes larger.

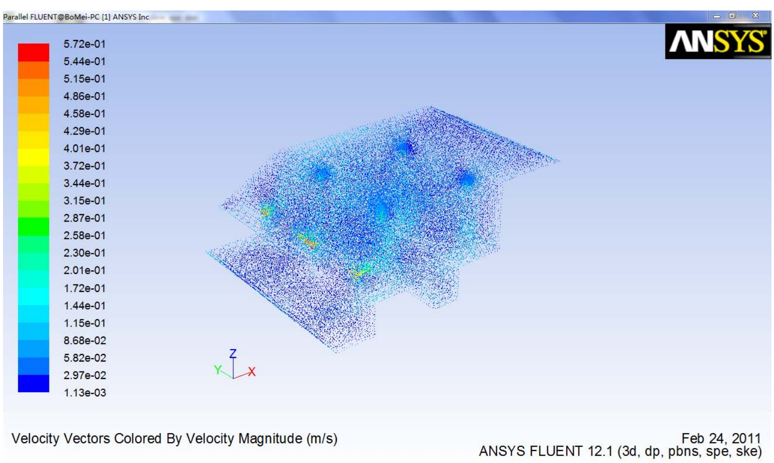

Figure 6: Velocity Vectors (Case 1)

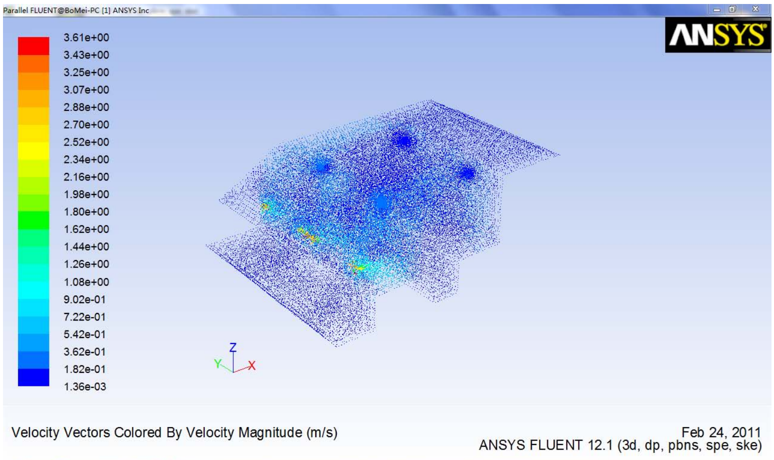

Figure 7: Velocity Vectors (Case 6)

2.5.2 *Carbon Dioxide Concentration*

Figure 4 and 8 through 12 show the concentration distribution of carbon dioxide at symmetry plane under different cases. After comparison, some comments are got. First, the bottom is much more red than the other areas in a domain. This means the concentration of carbon dioxide at the bottom is always the highest region. It is because carbon dioxide is species, which will be relatively significantly affected by gravity. Second, a region around the back row near the roof level becomes more and more red along with the increase of the fan speed. In other words, the region of high concentration of carbon dioxide at that area becomes more significant. Third, in case 1, the maximum is 2.91E-5 and the minimum is 1.26E-5. In case 6, the maximum is 1.54E-5 and the minimum is 1.20E-5. Both maximum and minimum values of case 6 are smaller than those of case 1. This represents that along with the increase of the fan speed, concentration of carbon dioxide will decrease as a whole. Forth, the difference between maximum and minimum is 1.65E-5 in case 1 and 0.34E-5 in case 6. The range of the scale in case 6 is smaller in the one in case 1, which means concentration of carbon dioxide will be more uniform as a whole along with the increase of fan speed. This is because for partially mixing in-door air quality modeling, mixing factor is small in case 1 and gets bigger in case 6.

3 Validations and Experimental Verification

For six cases, the CFD results are all converged after around 1000 iterations. The simulation is validated. The concentration of carbon dioxide at the symmetry plane,

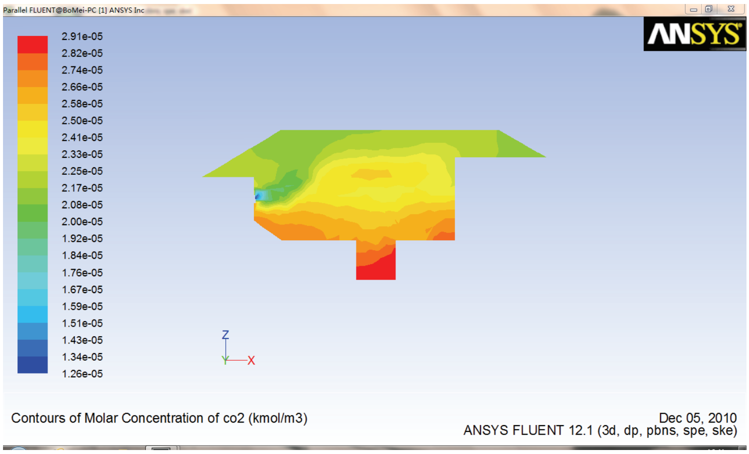

Figure 8: Concentration of Carbon Dioxide (Case 1)

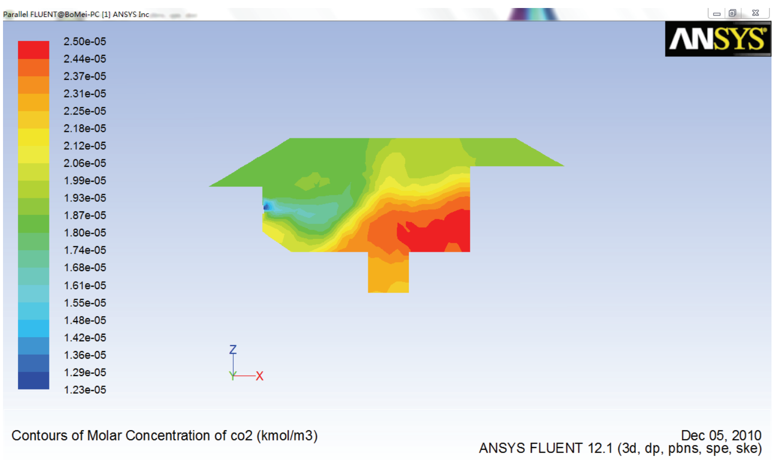

Figure 9: Concentration of Carbon Dioxide (Case 2)

between the two front seats, at head level is measured under six different cases, which is shown in Table 1.

4 Conclusions

First, concentration of carbon dioxide at the bottom is always the highest region among the whole domain. Second, along with the increase of the fan speed, a

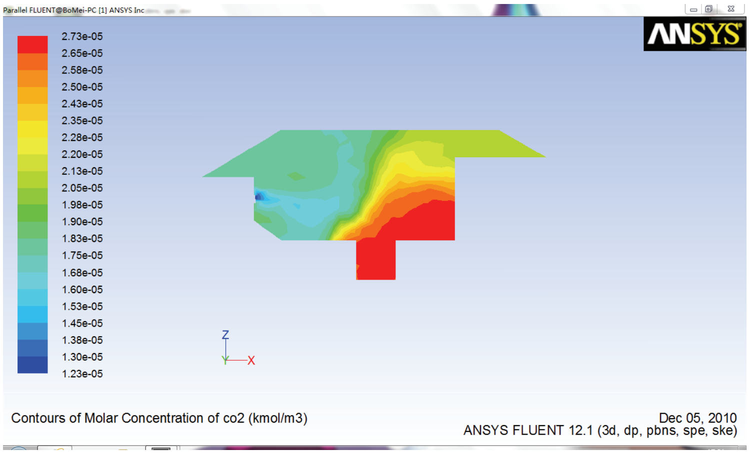

Figure 10: Concentration of Carbon Dioxide (Case 3)

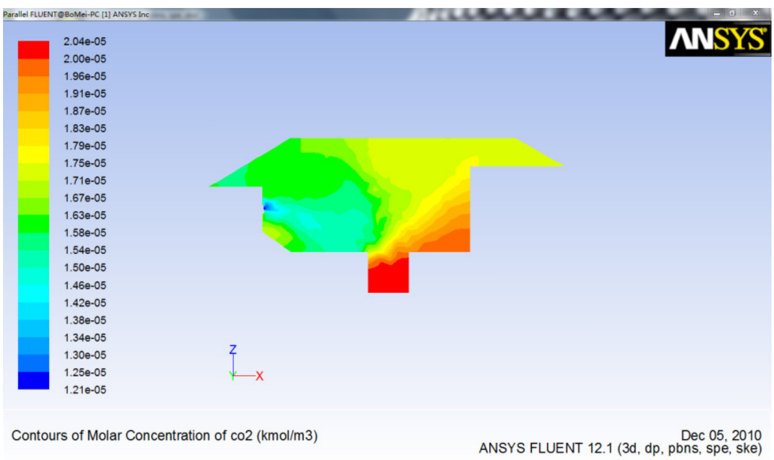

Figure 11: Concentration of Carbon Dioxide (Case 4)

region around the back row near the roof level with high concentration of carbon dioxide will become more and more significant. Third, along with the increase of the fan speed, concentration of carbon dioxide will decrease as a whole. Forth, along with the increase of the fan speed, concentration of carbon dioxide will be more uniform as a whole.

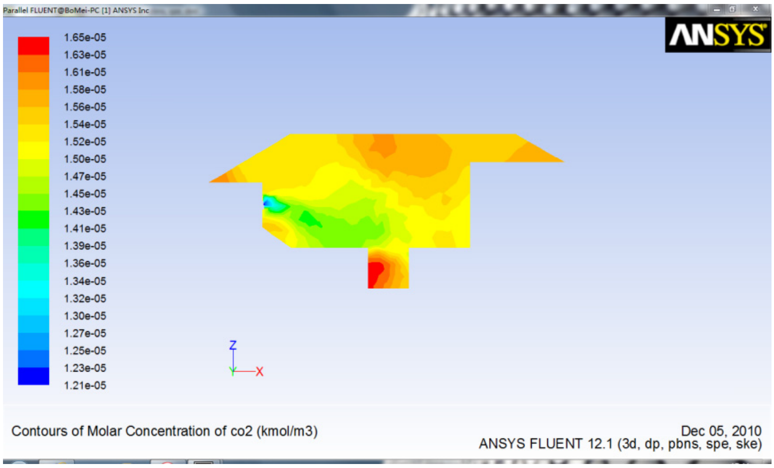

Figure 12: Concentration of Carbon Dioxide (Case 5)

Table 1: Error Level

Case	CFD Result	Experimental Result	Error Level
1	2.54e-5	26.2PPM	3.053%
2	1.84e-5	17.8PPM	3.371%
3	1.65e-5	17.0PPM	2.941%
4	1.56e-5	16.1PPM	3.106%
5	1.51e-5	14.5PPM	4.138%
6	1.45e-5	14.8PPM	2.027%
Average			3.106%

References

Zannetti, J. (2003): *Air Quality Modeling: Theories, Methodologies, Computational Techniques, and Available Databases and Software*. EnviroComp Institute and Air & Waste Management Association.

Patankar, S. V. (1980): *Numerical Heat Transfer and Fluid Flow*. Taylor & Francis Group, LLC.

Anderson, J. D. (1995): *Computational Fluid Dynamics: The Basics With Applications*. McGraw-Hill, Inc.

Rudolf, W. (1994): Concentration of air pollutants inside cars driving on highways and in downtown areas. *Science of the Total Environment*, vol. 146-147, pp. 433-444.

Rudolf, W. (1990): Concentrations of air pollutants inside and outside cars driving on highways. *Science of the Total Environment*, vol. 93, pp. 263-276.