Numerical Investigation of Air-Conditioned Flow Field in an Automobile Cabin with Parallel Unstructured Navier-Stokes?Solver

[1]Hua Chen
[1]Department of Information Communication, Leader Univ., Tainan, Taiwan
hchen@mail.leader.edu.tw
[2]Wen-Tai Chung and [3]Shen-Min Liang
[2,3]Institute of Aeronautics and Astronautics, National Cheng-Kung Univ., Tainan, Taiwan
p4893104@ccmail.ncku.edu.tw liangsm@mail.ncku.edu.tw

Abstract

As a result of the improvement of technology, the cabin equipments of automobiles are improved day by day. The thermal comfort in the automobile cabin has become important factor while consumers choosing new automobiles. Although computational fluid dynamics (CFD) has been developed to be a powerful designing tool for industrial applications, there are only few papers about air-conditioned flow field in the automobile cabin. In order to quantify the thermal comfort, Fanger [1, 2, 3] empirically established the well known PMV (predicted mean vote) equation by using parameters relevant to thermal comfort. The thermal comfort that a human feels in indoor environment has been known to be influenced mostly by six parameters: air temperature, radiation, air flow, humidity, activity level and clothing thermal resistance. This paper will focus on the flow velocity and temperature in the automobile cabin which are the key parameters in predicting thermal comfort.

Although three-dimensional simulation is feasible, complex geometries and high-cost computation power remain challenges. In order to overcome these difficulties, a parallel three-dimensional unstructured CFD code is used to simulate this problem. A cell-centered pressure-based method is adopted to simulate this problem. The method uses a second-order upwind scheme in space and a second order backward differencing scheme for time accuracy. The SIMPLE algorithm in conjunction with the Rhie and Chow [4] approach solves the Navier-Stokes equations, minimizing undesirable pressure oscillations. The turbulent model adopted in this study is F. R. Menter’s SST model [5]. In order to save space, the details of the cell-centered pressure-based method can be found in Chen, Huang and LeBeau [6, 7, 8].

Unstructured-grid CFD codes typically cost more in computational time than comparable structured-grid codes. This is because the difference of data structure between an unstructured-grid code and a structured-grid code. For an unstructured code, pre-fetching data through the cache is not as efficient as the cells, faces, and vertices tend not to be ordered in a sequential manner. Efficient parallel computation can compensate for this increased computational cost. Our parallel CFD code achieves this goal through a scheme that minimizes the information to be transferred. The highly portable message passing interface (MPI) protocol is implemented in a non-blocking fashion (using MPI_ISEND and MPI_IRECV) that masks the communication latency. In addition, specific MPI structures are used to pack different types of data into a single sending buffer.

Figure 1 shows the schematic diagram of the partitioning approach applied to the cell-centered pressure based method. Blue points indicate vertices, red points indicate

nodal points, and white points indicate the boundary points. By using this approach, the control volumes on the boundary are not split. Only communication of nodal values is needed for parallel computation which makes the implementation of MPI in an unstructured grid more straightforward.

Excellent load balancing between the subgrids on each node is achieved through using METIS [9] for domain decomposition. METIS can partition an unstructured grid into any integer number of zones without losing load balance. It is compatible with many platforms, convenient for running CFD codes on a variety of supercomputer to cluster architectures. The present partitioning approach has been tested by a number of two/three-dimensional geometries. All results show good load balances. The definition of load-imbalance rate LIMB and load-balance rate LB in this paper is defined as Eq. (1) and (2),

$$L_{IMB} = \frac{|N_{node} - N_{avg}|}{N_{avg}} \times 100\% \qquad (1)$$

$$L_B = 1 - L_{IMB} \qquad (2)$$

where N_{node} is grid size of the node and N_{avg} is the average grid size. By using load balance rate, we can compare the load balance quantitatively.

Figure 2 shows the load-balance distribution on each node with total 16 nodes tested. The x-axis indicates the node number and the y-axis indicates load-balance rate. The resulting load-balance rates are very close to 100% on every node with an average load balance rate of 98.37%. The present result is typical for the METIS-based partitioning approach, yielding good parallel performance.

For comparison, experimental results are also provided in this paper. The flow field in the automobile cabin is successfully simulated and the numerical results are in good agreement with the experimental data. The partitioned computational domain is shown in Fig. 3. Figure 4 shows the simulated temperature fields at different time steps. According to the simulated results, numerical simulations with turbulent model give more reasonable and accurate results in both flow and temperature field. The present results will be further used as the references for predicting thermal comfort in the automobile cabin.

Keywords: numerical simulation, automobile, air-conditioned, parallel computation

Reference

[1] Fanger, P. O., 밫hermal Comfort ? Analysis and Application in Environment Engineering,? Robert E. Krieger, Florida, 1982.

[2] Fanger, P. O. et al., 밃 ir Turbulence and Sensation of Draught,? Energy and Buidings, Vol. 12, 1988, pp. 21-39.

[3] Fanger, P. O. 밃 New Comfort Equation for Indoor Air Quality,? ASHRAE J., Vol. 31, 1989, pp. 33-38.

[4] Rhie, C. M. and Chow, W. L., 밡 umerical Study of the Turbulent Flow Past an Airfoil with Tailing Edge Separation,? *AIAA J.*, vol. 21, 1983, pp. 1525-1532.

[5] Menter, F. R., 밫wo-Equation Eddy-Viscosity Turbulence Models For Engineering Applications, *AIAA J.*, vol. 32, 1994, pp. 1598-1605.

[6] Chen, H., Huang, P. G. and LeBeau, R. P., 밣arallel 2D/3D Unsteady Incompressible Viscous Flow Computations Using an Unstructured CFD Code,? ICCFD3, Toronto,

Canada, July 2004.
[7] Chen, H., Huang, P. G. and LeBeau, R. P., 믧 Cell-Centered Pressure Based Method for Incompressible Unstructured Navier-Stokes Solver,? the 11th National CFD Conference, Tai-tung, Taiwan, August 2004.
[8] Chen, H., Huang, P. G. and LeBeau, R. P., 믧 Center Pressure Based Method for Two/Three- dimensional Unstructured Navier-Stokes Solver,? AIAA paper 2005-0880, AIAA Aerospace Sciences Meeting and Exhibit, Reno, NV, Jan, 2005.
[9] Karypis, G. and Kumar, V., A software package for partitioning unstructured graphs, partitioning meshes, and computing fill-reducing orderings of sparse matrices version 4.0 URL: http://www.cs.umn.edu/~karypis, [cited 1998].

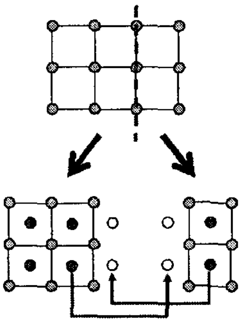


Fig. 1 A schematic diagram of cell-centered partitioning approach.

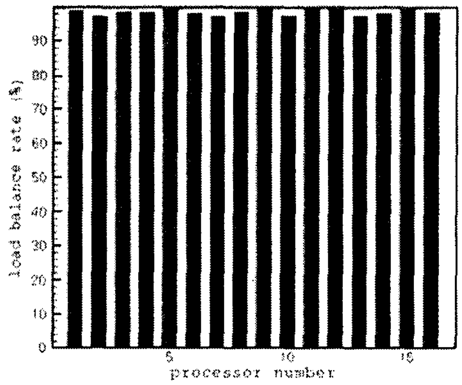


Fig. 2 Load-balance distribution on each node in parallel computation.

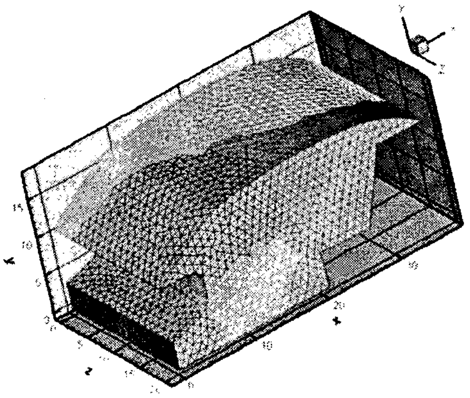
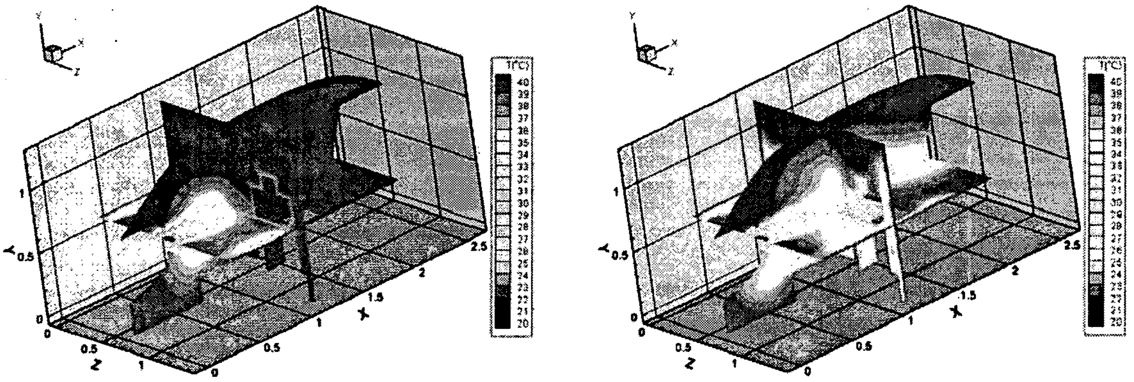
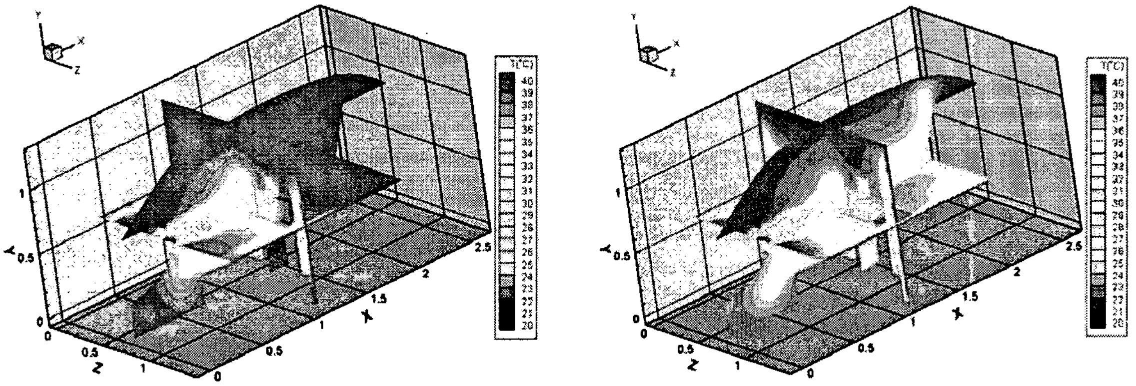


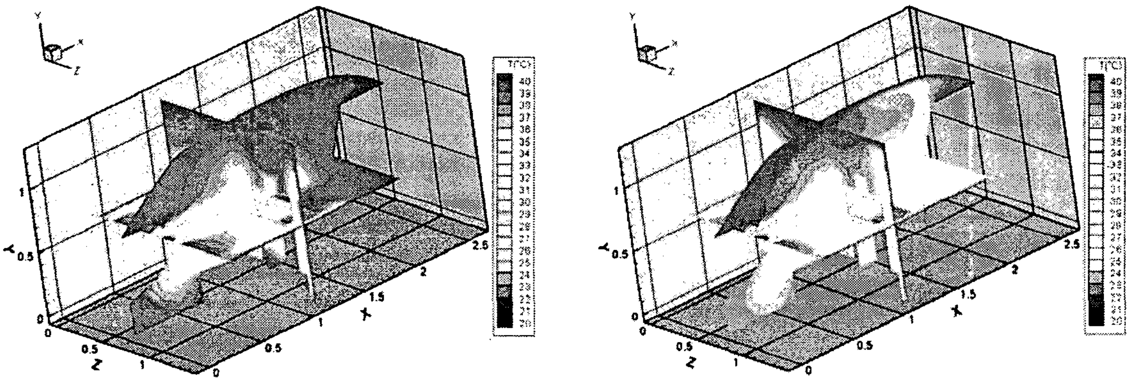
Fig. 3 The picture of partitioned computational domain.



(a) t=0.5 (d) t=2.0



(b) t=1.0 (e) t=3.0



(c) t=1.5 (f) t=4.0

Fig. 4 The simulated temperature fields at different time steps.