Numerical Simulation of the External Flow Field Around a Bluff Car*

Sun Yongling, Wu Guangqiang, Xieshuo

Automotive Engineering Department

Shanghai Tongji University

Shanghai, China

E-mail: wuqjuhyk@online.sh.cn

Tel: 021-65981770 (o), 021-65621602 (H)

ABSTRACT

Computational Fluid Dynamics, CFD, emerges as an advanced investigative means in the fields of automobile, aeronautics and aerospace. It reduces the wind tunnel experiments. In this paper, it is stated that CFD is applied to the automotive aerodynamics with Finite Element Method, FEM. MSC.Patran is used as a preprocessor and MSC.CFD as a solver. By that we get the pressure field and velocity field around the vehicle, then acquire air drag coefficient (Cd) and compare it with the wind tunnel experimental results. The conclusion of the comparison is satisfying.

Key Word: CFD, external flow field, Car

Introduction

With the increasing improvement of the automobile and freeway technique and the decrease of petrol resource, people ask for higher automotive tractive ability, lower fuel consumption and better handling. Automotive aerodynamics characteristics are very important to these capacities. At present, wind tunnel is a chief research means in our country, but Computational Fluid Dynamics, CFD, has developed very fast in some countries, with the appearance of high performance computer and accurate analytic method. As a modern method, CFD can not only shorten the automobile design period, but also greatly reduce the wind tunnel experiments.

In this article, we study a certain car. In order to obtain the accurate result with less time and cost, we adopt simplified model of the car, which can efficiently stand out the main facets of the problem. On the CAD platform such as Unigraphics (UGII) the geometrical model is set up, then is imported into the preprocessor software MSC.Patran. After meshing it, we get the finite element model, which is exported into MSC.CFD to proceed fluid analysis. On the base of analysis result it can display the pressure contour and the velocity vector of the external field of the body. By the postprocessor we acquire the air drag coefficient (Cd).

Geometrical Model

On the platform of UGII, the geometrical model is created (see Figure 1). The basic parameter of the model is 4246×1402×1036(mm) and the tire is neglected. To compare the results with those of experiment in TJ-2 wind tunnel, the calculating region is defined as 60×12×10 (m).

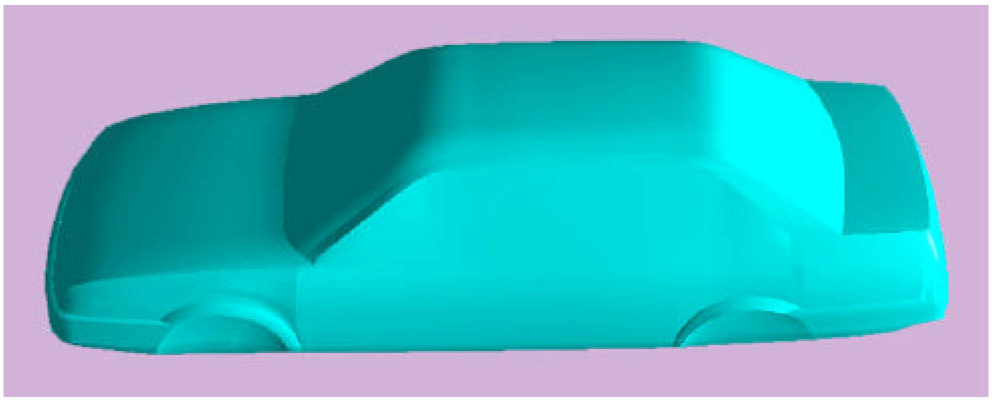

Fig. 1 Geometrical Model

Finite Element Model

The output format of the geometrical model is Parasolid, which can be read into MSC.Patran. We apply Paver method to the complex fluid region, by selecting triangle plane cell (Tri3) and tetrahedron cell (Tet4). At first we mesh the boundary surface of the computational field, that is to say we create the finite element grids on six planes of the calculating space and the exterior surface of the body, then the whole computational field is meshed with Tet4 automatically. There are total 52118 elements. To improve calculation precision and accuracy, the grids of the body surface are denser, and that of far off fields are coarser.

Boundary Conditions

The boundary conditions are as follows:

- Inlet conditions, the velocity of coming flow is given as V_∞=30m/s the turbulent intensity is 1%.
- Outlet conditions, the gradient along main flowing direction is zero, the standard atmospheric pressure is put on exit boundary face.

The top, bottom, left and right face of the computational region are considered as walls. This is simulated by Wall Function of MSC.CFD.

Calculation Results and Analysis

Three-dimension constant incompressible viscid N-S equation is used as governing equation and standard κ-ε model is applied to simulate turbulent computation. Figure2 and 3 show the pressure contour and the velocity vector on the symmetric section of the model, respectively.

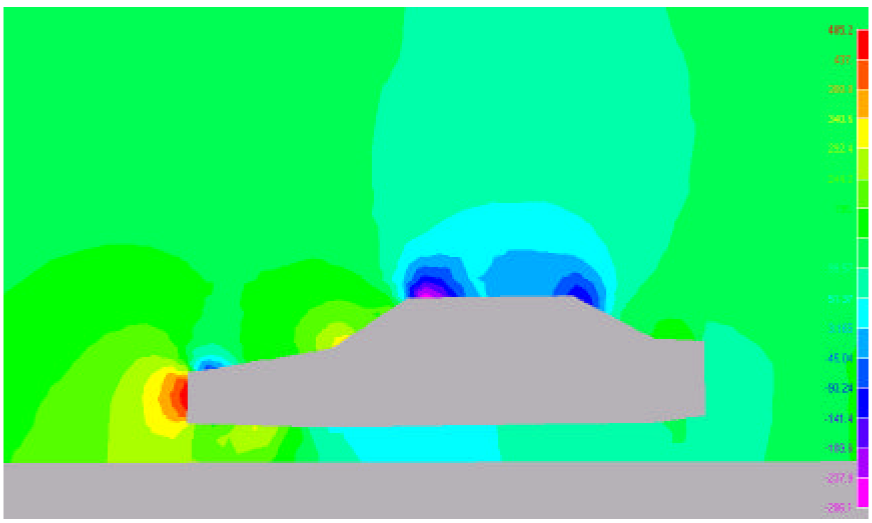

Fig. 2 Pressure Contour

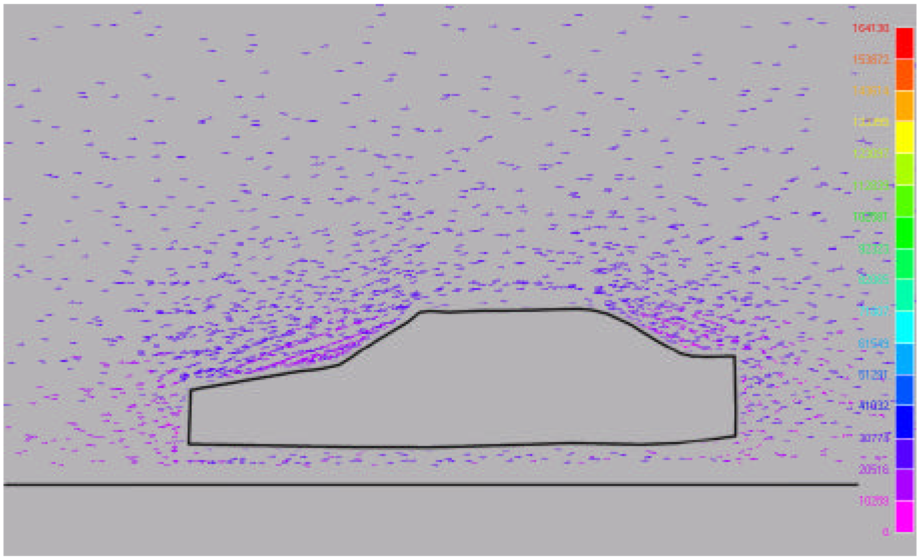

Fig. 3 Velocity Vector

In the pressure contour, it can be easily found that there are positive pressure areas at the front of the body, especially in the area before the radiator and the boundary area between hood and the front windshield. However negative pressure area is also found at the front end of the hood. Both the front and rear end of the roof are negative pressure areas. Those results agree to the experimental results in TJ-2 wind tunnel. There are no obvious negative pressure areas at the tail of the model, which is different from the wind tunnel result. This shows that the model should be amended.

In the velocity vector, it can be seen that there are small separation sectors in the areas above the radiator, near the bumper, and above the front and rear windshields. And some eddy flow is found in the wake of the body. In the area near to the symmetric sectio n, air flow is parallel to ground while it is apart from the roof, and slightly deflects to both sides. Air between floor and the ground goes upwards while leaving from the body. The main direction of the wake is upward. In the areas near to the sides of the body, air from the bottom is whirling and a large eddy is found (shown in figure 4). air from the roof of either side whirls softly, it forms rather small eddies, which are not obvious.

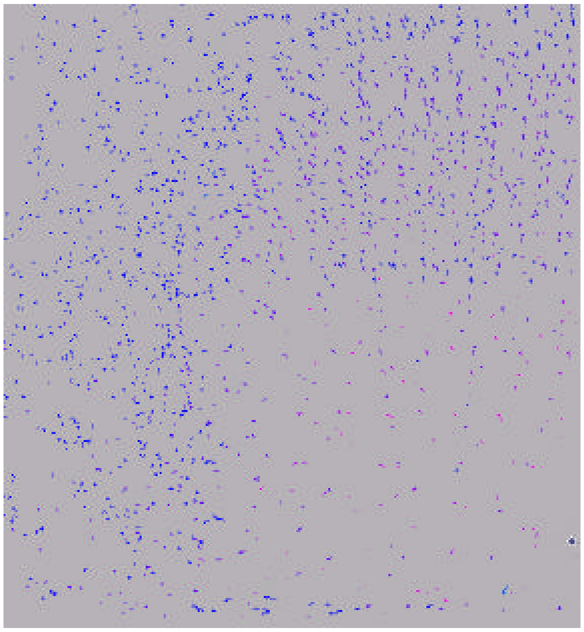

Fig. 4 Velocity Vector in the Wake

Those feathers of external filed are proved by experiments. Based on the output file of MSC.CFD, the drag coefficient of this model is calculated, C_D=0.347. The experimental result is 0.364 (1/4 wind tunnel model). The calculation result is less than experimental result, which is resulted from many factors, such as the geometrical model is not precise enough, boundary layers of the ground and body are ignored, and additionally the model doesn't take the effect of the internal flow into account.

Conclusion

From the above discussions, we can conclude that MSC.CFD is a good platform to simulate flow filed, the method proposed in this paper is suited for the calculation and analysis of automotive external flow field.

References

1) Mark P.Miller. Getting Started with MSC/NASTRAN, User's Guide, The MacNeal-Schwendler Corporation, 1996.
2) User's Manual, Volume 2, MSC/PATRAN V77, The MacNeal-Schwendler Corporation, 1997.
3) CFDesign Tutorial Manual, Blue Ridge Numerics, Inc., 1992-1998.
4) CFDesign Solver Technical Reference, Blue Ridge Numerics, Inc.1992-1998.
5) CFDesign Solver User's Guide, Blue Ridge Numerics, Inc.1992-1998.
6) Zhang Hualin, Aerodynamics Simulation of External Flow of Automobiles, Automobile Engineering, 2000, 117-120.
7) Fu Limin, Aerodynemics in Automobile, Publication House of Mechanical Engineering, 1997.