www.cadanda.com
Computer-Aided Design AND APPLICATIONS

Analysis of Flow over a Convertible

Aniket A Kulkarni[1], S V Satish[2] and V Saravanan[3]

[1]PES Institute of Technology, India, aniket260919902@gmail.com
[2] PES Institute of Technology, India, svsatish@pes.edu
[3] PES Institute of Technology, India, Saravanan.venkatesh@gmail.com

ABSTRACT

Analysis of the flow over automobiles using Computational Fluid Dynamics (CFD) was initially introduced in F1 racing event which helped in modifying the design of the racing cars to get the maximum down force and least possible drag. Later this method of analysis was used in different segments of cars to optimize the design, so that, improvement in the fuel-efficiency and other performance parameters was achieved. This presentation will highlight the analysis of flow over a convertible; it involves comparison of the coefficient of drag for two configurations of the convertible- with the roof and without the roof. The work involves creating a 3D model of the convertible and meshing it using Gambit. Next step is importing it in FLUENT and analyzing the flow using k-epsilon turbulence model.

Keywords: CFD analysis, convertible, ANSYS Fluent.
DOI: 10.3722/cadaps.2012.PACE.69-75

1 INTRODUCTION

For a moving vehicle, drag refers to the forces that oppose the relative motion of the vehicle through air. The measure of these forces is the coefficient of drag which is a dimensionless quantity. The coefficient of drag and the drag forces are in directly proportional to the speed of the vehicle. In our paper, we will conduct the analysis of flow over a car with two configurations; one with the roof and the other without one. Flow of air near the body is modeled using the relationship between the pressure and velocity expressed by Bernoulli's equation:

$$P_{static} + P_{dynamic} = P_{total}$$

Or,

$$P_s + 1/2\rho v^2 = P_t$$

Where,

ρ = air density (kg/m^3)

v = relative air velocity (m/s)

As the speed of air along the car body decreases to zero, it results in variation in pressure. This phenomenon determines the force of drag, caused by interaction of air. This force, known as drag is given by:

$$D = 1/2 \rho v^2 \times C_D \times A$$

Where,

C_D = coefficient of drag

A = reference area of the car

The reference area of the car can be found out as

$$A = B \times H$$

Where,

B= breadth of the car

H= height of the car

2 MODEL CREATION

For the present paper we have chosen two simplified models with same major dimensions. The reference values of length, height and breadth are taken from the internet.
After collecting all the dimensions, the models were created using Autodesk Inventor software. These models were meshed using Gambit. The following figures show the models that were created:

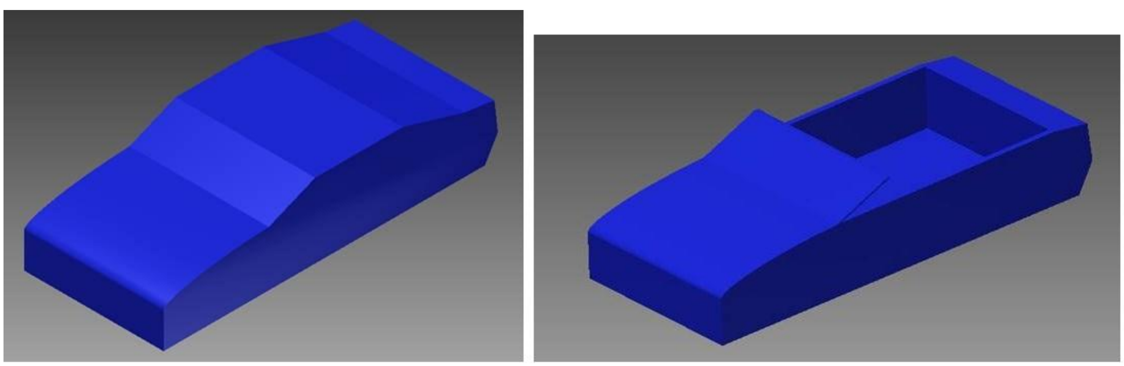

Fig. 1: The model that were created using Autodesk Inventor. This kind of a car is usually referred to as a convertible: (a) Convertible with the roof closed, (b) Convertible with the roof open.

Version	Sedan	Convertible
Length(mm)	2093	2093
Width(mm)	867	867
Height(mm)	461	461

Tab. 1: Major dimensions of the two variants of the car.

3 NUMERICAL SIMULATIONS

There are two ways to analyze the models to study the aerodynamic behavior of the car- wind tunnel and numerical simulations. Because of its proven efficiency and financial reasons, the latter one is preferred. As mentioned before, preprocessing of mesh generation was done in Gambit and processing was done in Fluent.

3.1 Numerical Simulation Methods

The governing equations based on conservation of mass, momentum and energy are solved for each of the element. Fluent uses finite volume method to solve the governing equations. The flow is said to be turbulent when all the transport quantities, mass, momentum and energy exhibit periodic, irregular fluctuations in time and space. There are a lot of solution methods or models available in Fluent. Based on required accuracy, available hardware resources and computational time, the model is to be selected. We have used the k-epsilon turbulent model for our analysis, since, it's very commonly used. It's semi-empirical, two equation based model, which means that it includes two extra transport equations to represent the turbulent characteristics of the flow. The first transported variable is the turbulent kinetic energy k and the other is the turbulent dissipation ε (epsilon). ε determines the scale of turbulence and k determines the energy involved in turbulence.

The transport equations for a standard k-epsilon model are:

1. $\partial/\partial t(\rho k) + \partial/\partial x(\rho k u) = \partial/\partial x[(\mu+\mu/\sigma_k)\,\partial/\partial x(k)] + G_k + G_b - \rho\varepsilon - Y_M + S_k$
2. $\partial/\partial t(\rho \varepsilon) + \partial/\partial x(\rho \varepsilon u) = \partial/\partial x[(\mu+\mu/\sigma_\varepsilon)\,\partial/\partial x(\varepsilon)] + C_{1\varepsilon}\,\varepsilon/k(C_k+C_{3\varepsilon}G_b) - \rho\, C_{2\varepsilon}\,\varepsilon^2/k + S_\varepsilon$

In these equations, G_k represents the generation of turbulent kinetic energy due to mean velocity gradients. G_b is the generation of turbulent kinetic energy due to buoyancy. Y_M represents the contribution of the fluctuating dilatation in compressible turbulence to the overall dissipation rate. $C_{1\varepsilon}$, $C_{2\varepsilon}$ and $C_{3\varepsilon}$ are constants; σ_k and σ_ε are the Prandtl numbers for k and ε respectively. S_k and S_ε are user-defined source terms.

4 VIRTUAL WIND TUNNEL CREATION

After creating the basic shape of the car, we went on to create a virtual wind tunnel. It is a box shaped structure which encloses the car. The entrance of the wind tunnel is placed at a distance equal to the length of the car and this will be the face where air enters the wind tunnel. The exit of the wind tunnel is placed at a distance equal to five times the length of the car. This face represents the region where the air gets out of the wind tunnel. The width of the wind tunnel is almost equal to the length of the car. The mesh was generated in Gambit. There are 10 lakh elements in the mesh that was generated.

The figure below gives a good representation of the wind tunnel and the mesh generated:

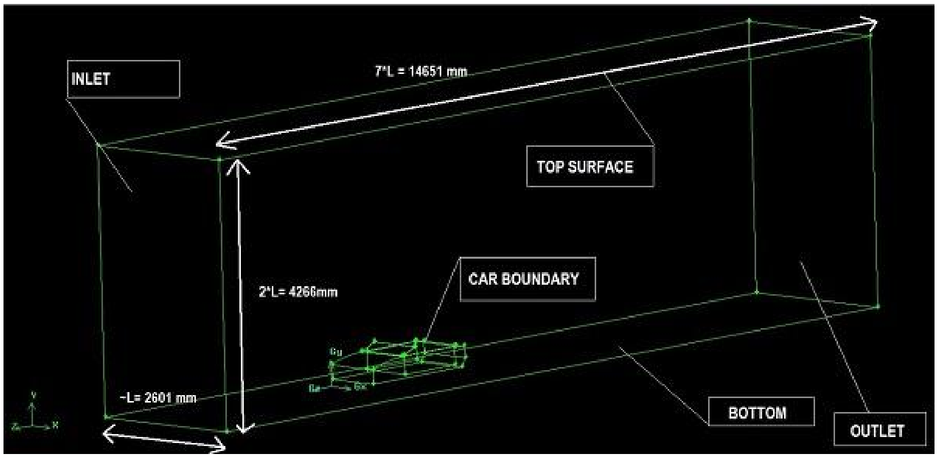

Fig. 2: virtual wind tunnel.

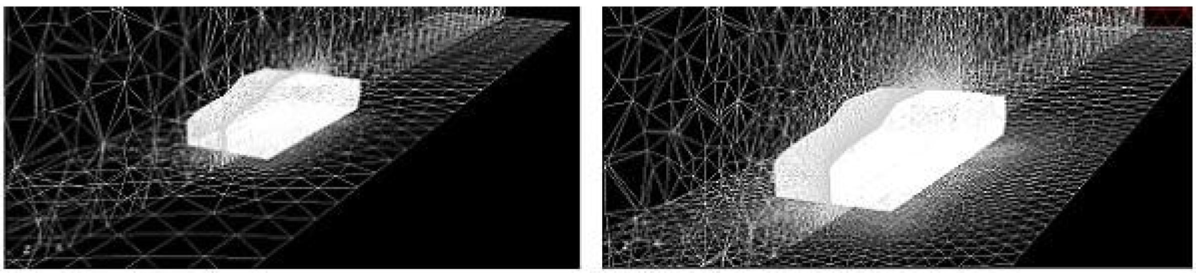

Fig. 3: A general comparison of the meshes that were generated for the analysis.

The exit part of the car which contains the rear part of the car is given a lot of space in order. This will make sure that the flow behind the car is completely captured, because, this is the region where the changes in the flow are expected to occur and results are obtained.

5 BOUNDARY CONDITIONS

The other boundary conditions are set as shown in the table below:

Boundary	Boundary Conditions
Car surface	Wall
Inlet	Velocity Inlet
Outlet	Pressure Outlet
Side walls	Wall
Top and Bottom surfaces	Wall

Tab. 2: List of Boundary conditions.

For our simulations, we have considered the velocity of air to be equal to 55.5 m/s, which is equal to 200 km/h. The Reynolds number for our flow is calculated as:

$$R_e = \rho \times v \times L/\mu$$
$$=1.225 \times 5.55 \times 2.093/1.7894 \times (10^{-5})$$
$$=3.3 \times e+06.$$

The Reynolds number thus calculated corresponds to turbulent flow. This justifies our use of k-epsilon model for our simulation.

6 RESULTS

The following figures represent the velocity contours for both the configurations of the car at the mid-plane.

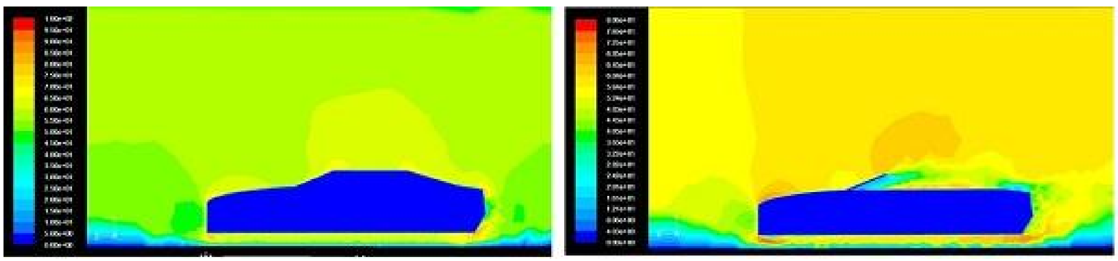

Fig. 4: velocity contours for the flow over convertible: (a) with roof closed, (b) with the roof open.

From the above figures, we can see that the velocity is almost zero near the rear of the convertible with the roof open. This zone represents the zone of maximum turbulence.

From figures 4, 5 and 6, it can be seen that, air slows down in the front part of the car as more number of molecules are accumulated into smaller space. Once air comes to stagnation state, a lower pressure area is required. The top and side regions correspond to this condition.

Below given are the static pressure contours of the car at the mid plane of the car:

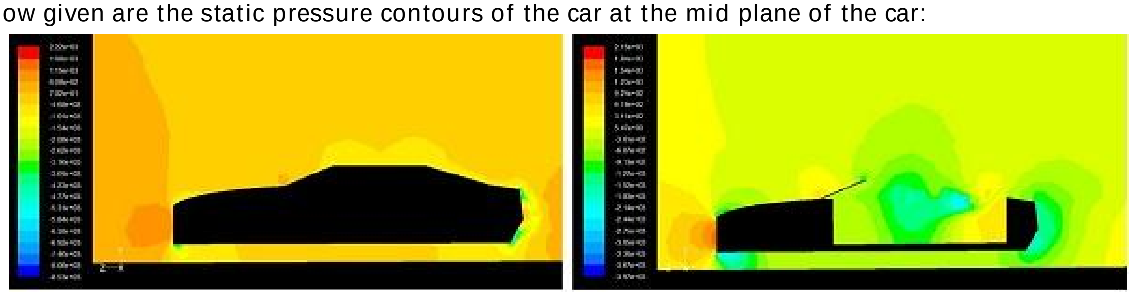

Fig. 5: pressure contours for the flow over convertible: (a) with roof closed, (b) with the roof open.

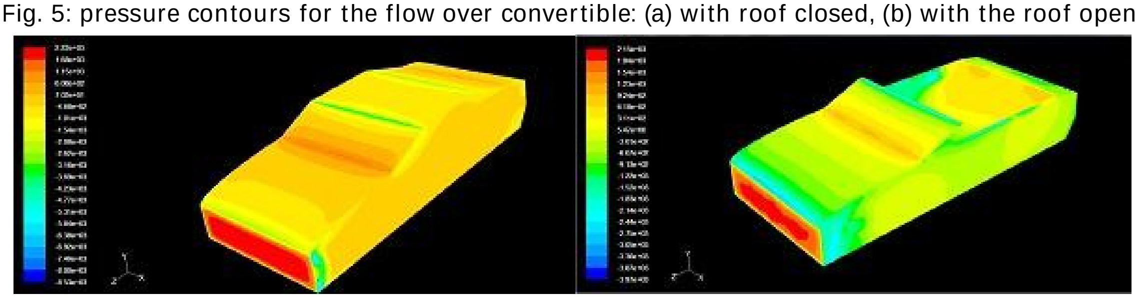

Fig. 6: Pressure contours-isometric view (a) with roof closed (b) with roof open.

As the air flows over the car hood, pressure is decreasing, but when it reaches the front windshield it briefly increasing. When the higher pressure air in front of the windshield travels over the windshield, it accelerates, which causes decrease in pressure. This lowering of pressure creates lift force on the car roof as the air passes over it.

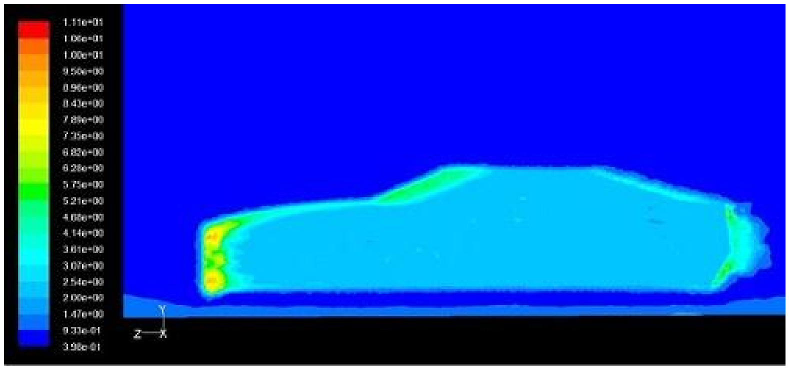

Fig. 7: turbulence intensity for convertible with roof closed at the mid-plane.

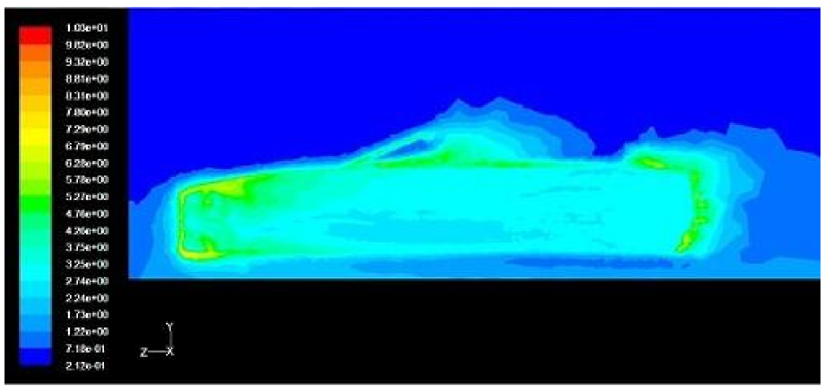

Fig. 8: turbulence intensity for convertible with roof open at mid-plane.

A comparison of the coefficient of drag is indicated in the table below:

Version	C_d - Coefficient of drag
Convertible with roof closed	0.75
Convertible with roof open	0.97

Tab. 3: comparison of drag coefficients.

7 CONCLUSIONS

The pressure at the back of the car, in both the cases is higher than the front, which contributes into drag force, called form drag. Near the body of the car, a thin boundary layer exists, where the air speed is reduced to zero. If the boundary layer is attached to the car, then very low drag coefficient is obtained, but if the boundary layer separates, the drag coefficient usually increases (for the case of convertible with roof open).

The aim of this paper was to compare the values of the coefficient of drag between the two versions of the convertible- roof open and roof closed. From the above table, it can be seen that the value for convertible with roof open is higher than that for the convertible with the roof closed. From the literature obtained from the internet, the values follow the trend that the coefficient of drag is higher for the convertible with roof open.

With the kind of geometry used in the analysis, numerical simulation level and number of elements used in the mesh the obtained results are satisfactory.

As a future work we will try to conduct the analysis with models having minute details, for specific regions where turbulence is found to be high (at the rear region of the car) and with more number of elements in the mesh.

8 REFERENCES

[1] Fluent 6.2.16 userôs guide

[2] Gambit 2.2.30 userôs guide

[3] http://www.grabcad.com

[4] John D Anderson, Jr.: Computational Fluid Dynamics, The Basics with Applications, McGraw-Hill Inc., 1995.

[5] Katz, J.: Race car aerodynamics, Bentley Publishers, 1995.