External Flow Analysis over Hyundai Model Using CFD

Assist.Proff.Karim Hassan Ali, Dr.Abdessamed Kacem

[1] Baghdad University / Engineering College
[2] Baghdad University / Engineering College
Corresponding Author: Assist. Proff.Karim Hassan Ali

Abstract: *Computational fluid dynamic software ANSYS Fluent 15.0 is used to predict the physical parameters on the external surface of a vehicle. The simulation of the wind tunnel experiments using the above CFD package would be carried out. To study the aerodynamic influence of vehicle segments are considered as three-box form passenger car Hyundai model. As this model provided with the extremities in the design of vehicle category in summarizes these results in a broader sense to understand the aerodynamics of such profile. A wind tunnel experiments consists of physically testing the model in a simulated environment. It is a closed environment in which air is blown across the model at a constant velocity. As the air passes over the vehicle, it follows the streamlines pattern present on the body. This causes a change in pressure and velocity characteristics across the model. By studying the pressure and velocity variations across the model, the drag and lift coefficient could be predicted.*
Key words: *CFD, External flow field, Three-box form car, Drag coefficient, Lift coefficient.*

I.   Introduction

An accurate vehicle model an accounts are related to the accuracy in the velocity and pressure variations across the body during testing. To obtain correct results, a true representation of a vehicle is tested in a simulated wind tunnel experiment [1]. In real world, full size vehicle models are tested in a wind tunnel to obtain its behavior as shown in figure (1).

II. Grids And Boundary Conditions

Vehicle model is created using ANSYS Fluent 15.0. This model is an approximation to the realistic vehicle and considered for the numerical (CFD) analysis, three-box form to a study the external flow characteristic and the effect of vehicle body profile on it. The computational domain is discredited into 448564 tetrahedral elements for three-box form, as shown in figure (2). As shown in figure (3), the vehicle is placed approximately in the middle of the numerical model of the wind tunnel in the length as well as in width. There is a ground clearance in between the floor the vehicle and the bottom wall of the wind tunnel. The tunnel wall and the vehicle body are considered as typical wall boundary conditions. At the inlet of the wind tunnel, a constant mean velocity of 45 m/s or 160 km/h is applied and the outlet of the wind tunnel is considered as an outflow boundary. For solving the flow and turbulence parameters, a three dimensional finite volume numerical method and a standard k-ε turbulence model based on commercial code ANSYS Fluent 15.0, are used for analysis.

III. Numerical Formulation

For solving the standard variables u, v, w, k, ε and T, three-dimensional Navier-Stokes equations for steady, turbulent, and incompressible flow [2] can be written as:

1-3-1 Governing Equations

Mass (continuity) equation:

$$\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x_i}(\rho u_i) = 0 \tag{1}$$

Momentum equation:

$$\frac{\partial}{\partial t}(\rho u_i) + \frac{\partial}{\partial x_j}(\rho u_i u_j) + \frac{\partial P}{\partial x_i} - \frac{\partial \tau_{ij}}{\partial x_j} = 0 \tag{2}$$

Energy equation:

$$\frac{\partial}{\partial t}(\rho H) + \frac{\partial}{\partial x_j}(\rho u_i H - u_i \tau_{ij}) = 0 \tag{3}$$

Where the stress tensor τ_{ij},

$$\tau_{ij} = \left[\mu(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i})\right] - \frac{2}{3}\mu\frac{\partial u_j}{\partial x_i}\delta_{ij} \tag{4}$$

1-3-2 Turbulence modeling

The standard k-ε turbulence model [3] is used which contain two equations one for turbulence kinetic energy and the other for its dissipation rate obtained from the following equations respectively:

$$\rho\frac{\partial k}{\partial t} = \frac{\partial}{\partial x_i}\left[(\mu + \frac{\mu_t}{Pr_k})\frac{\partial k}{\partial x_i}\right] + (2\mu_t\delta_{ij} - \frac{2}{3}\rho k\delta_{ij})\frac{\partial u_j}{\partial x_i} - \rho\varepsilon \tag{5}$$

And….

$$\rho\frac{\partial \varepsilon}{\partial t} = \frac{\partial}{\partial x_i}\left[(\mu + \frac{\mu_t}{Pr_k})\frac{\partial \varepsilon}{\partial x_i}\right] + C_{1\varepsilon}\frac{\varepsilon}{k}(2\mu_t\delta_{ij} - \frac{2}{3}\rho k\delta_{ij})\frac{\partial u_j}{\partial x_i} - C_{2\varepsilon}\rho\frac{\varepsilon^2}{k} \tag{6}$$

In the above equation, the last three terms on the right hand side shows the diffusion generation, and the dissipation rates of ε and the turbulent viscosity derived is,

$$\mu_t = C_\mu\rho\frac{k^2}{\varepsilon} \tag{7}$$

Where $C\mu=0.09$

The simulation results are obtained under the convergence criteria of 1.1^{-10} for the residuals of u, v, w, k and ε. The drag coefficient can be calculated using the following equation.

$$C_D = \frac{F_D}{\frac{1}{2}\rho_\infty V_a^2 A_f} \tag{8}$$

Where F_D is the force acting in the direction of air flow on the frontal area of the vehicle body and ($\frac{1}{2}\rho_a V_a^2$) is free stream dynamic pressure. In the present solution, a semi-implicit method for pressure linked equation (SIMPLE) is used. Figure (4) shows the flow chart for external flow modeling that illustrates the options used in ANSYS Fluent 15.0 software.

IV. Results And Discussion

Figure (5) shows the velocity increasing smoother when compared to that of the wagon. This is attributed due to the smooth profile changes across the hood and the roof. There is a subtle change in velocity at the rear end as the flow separation occurs [4]. Figure (6) shows the velocity vectors colored by velocity magnitude (m/s). Figure (7) shows Pathlines Colored by Particle [5]. Figure (8) shows Pathlines Colored by Velocity Magnitude (m/s). Figure (9) shows Countours of Velocity Magnitude (m/s). Figure (10) shows Air drag coefficient for Hyundai model with a value of 0.53. Figure (11) shows Lift coefficient for Hyundai model with a value of -0.65.

V. Conclusion

Through aerodynamic numerical simulation the drag coefficient are gotten and some advices to reduce the aerodynamic drag could be brought out too. It is a simple, effective, convenient and fast way to do aerodynamic numerical simulation based CFD in the process of car styling. In the case of three box form model, low-pressure pockets are present at the top and bottom of the vehicle. This differential pressure is almost negligible that keeps the vehicle on ground.
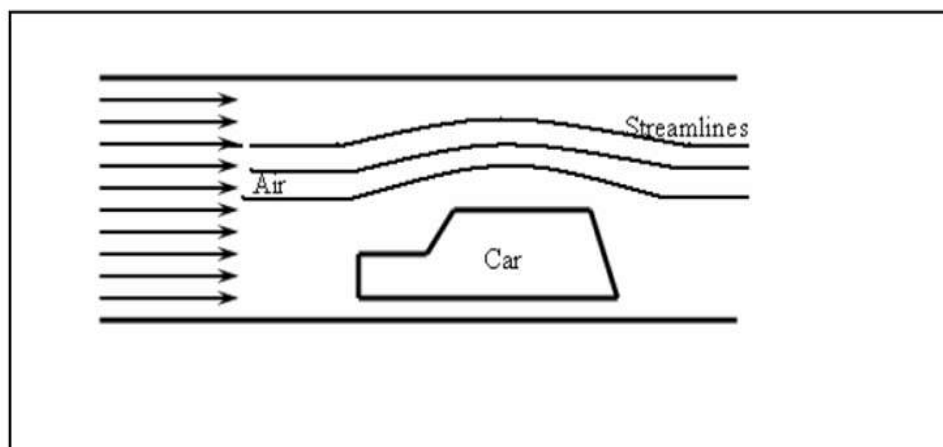
Notations
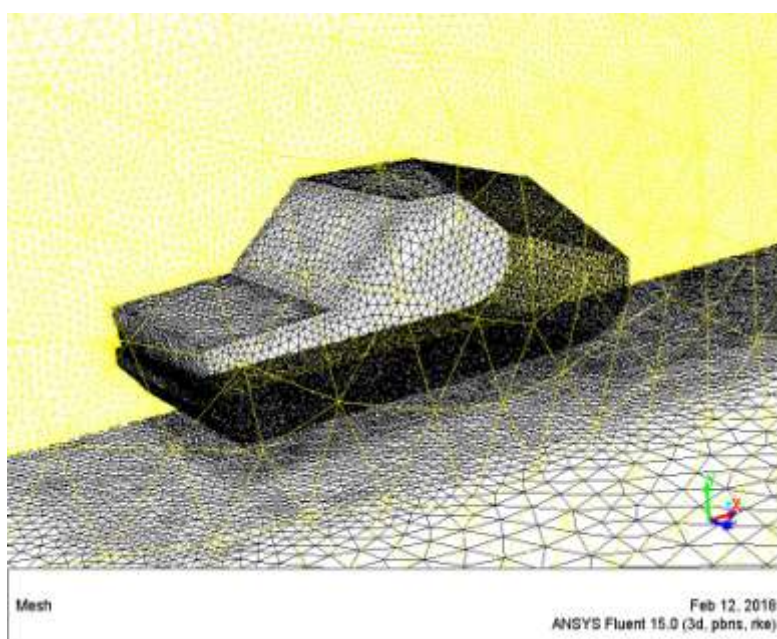A: The model frontal area (m^2).
C_D: Drag coefficient.

F_D : Total drag force(N).

V_∞: Undisturbed air flow (m/s).

g : Gravitational acceleration (m/s^2).

ε : Dissipation rate.

ρ : Density(Kg/m^3).

σ : Normal stress.

ν : Kinematic viscosity.

μ_t : Additional viscosity due to turbulence.

τ_{ij} : Stress tensor.

References

[1]. Aerodynamic Testing of road vehicles- Testing methods and procedures (Stabilized Apr 2016).
[2]. Turbulence models and their applications, T.S.D.Karthik, 10th Indo German winter Academy 2011.
[3]. Philip J. Pritchard, Fox and McDonald's, "Introduction to Fluid Mechanics", 2011.
[4]. Yingchao Zhang , Zhe Zhang, Shuanghu Luo and Jianhua Tian, "Aerodynamic Numerical Simulation in the Process of Car Styling", Applied Mechanics and Materials Vols. 16-19 (2009) pp 862-865.
[5]. Akshay Parab, Ammar Sakarwala, Bhushan Paste, Vaibhav Patil and Amol Mangrulkar, " Aerodynamic Analysis of a Car Model using Fluent- Ansys 14.5", International Journal on Recent Technologies in Mechanical and Electrical Engineering (IJRMEE),Volume: 1 Issue: 4, November 2014.

Figure(1): Wind tunnel testing

Figure(2): ANSYS Fluent 15.0 for Hyundai model

Model length = 3970 mm
Model Height = 1360 mm
Model Width =1558 mm
Domain Height = 5000 mm
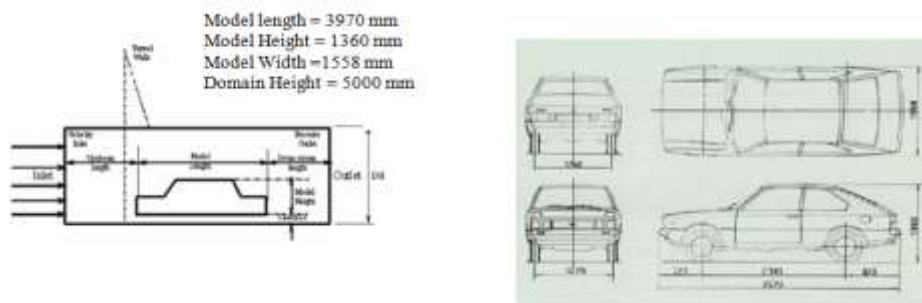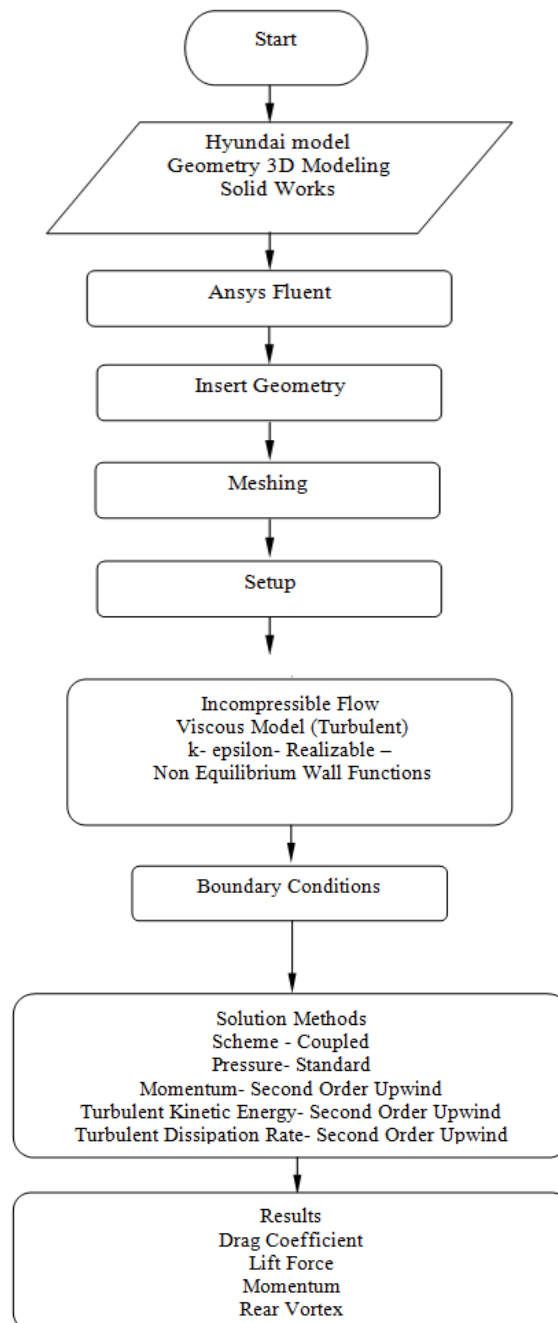
Figure(3): Three- box form (Hyundai) car model

Figure (4): Shows the flow chart for the external flow modeling
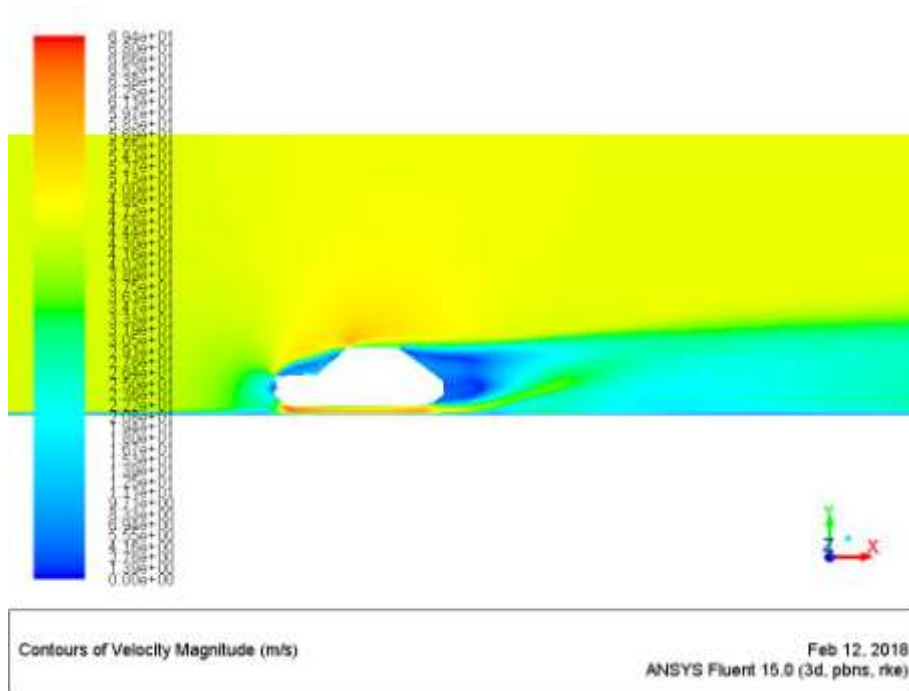That illustrates the options used in ANSYS Fluent 15.0

Contours of Velocity Magnitude (m/s)

Feb 12, 2018
ANSYS Fluent 15.0 (3d, pbns, rke)

Figure (5): Velosity vectors on the surface of vehicle (m/s)
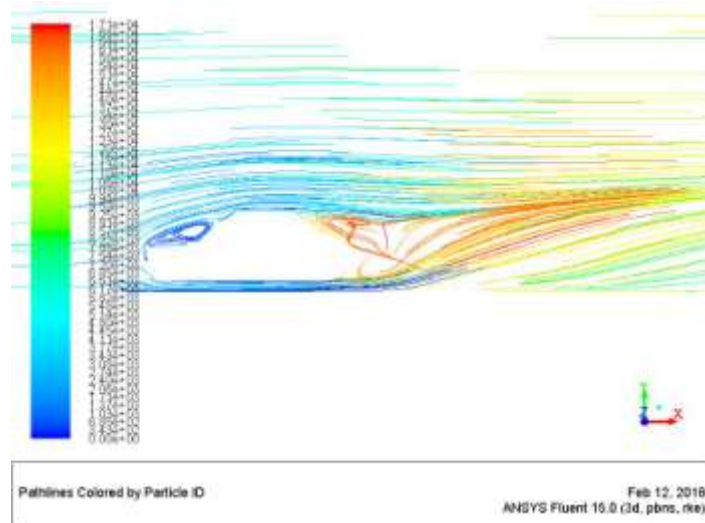

Velocity Vectors Colored By Velocity Magnitude (m/s)

Feb 12, 2018
ANSYS Fluent 15.0 (3d, pbns, rke)
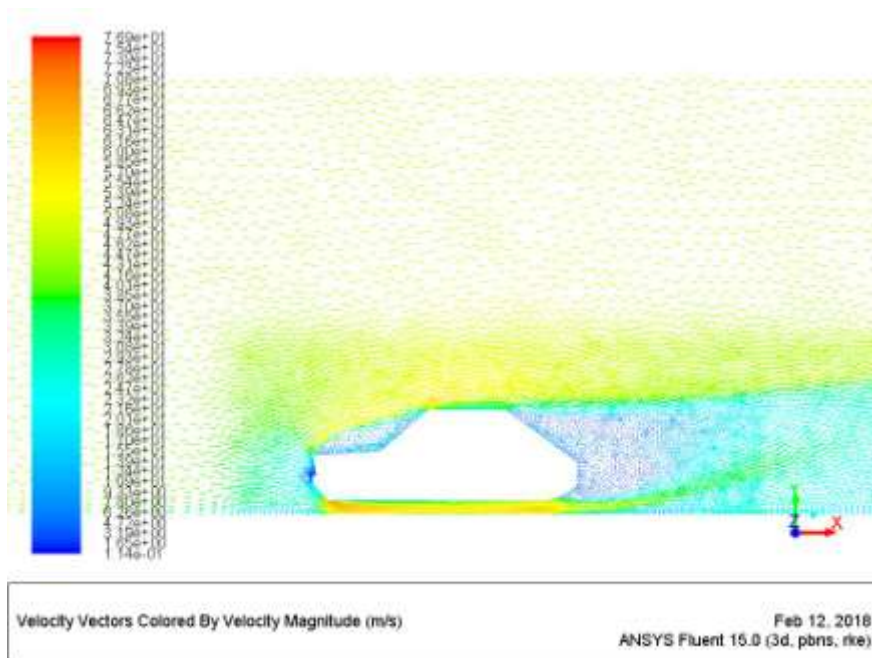
Figure (6): Velocity vectors colored by velocity magnitude (m/s)

Figure (7): Pathlines Colored by Particle
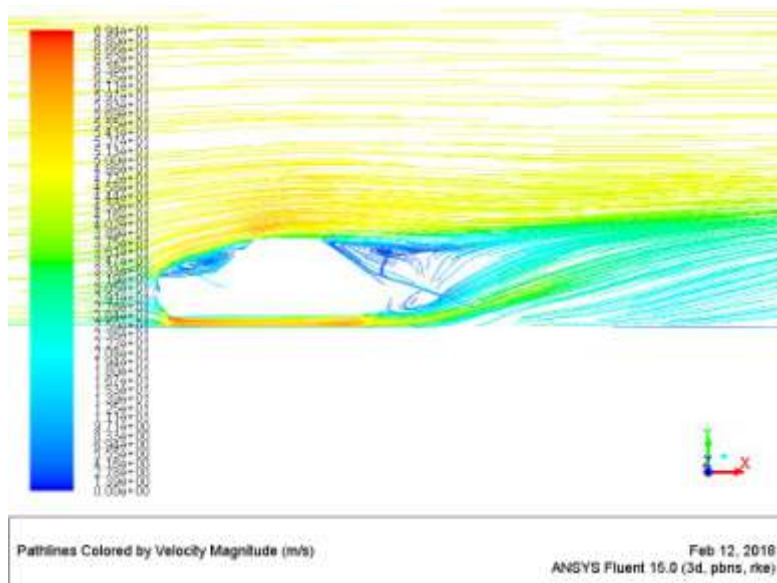
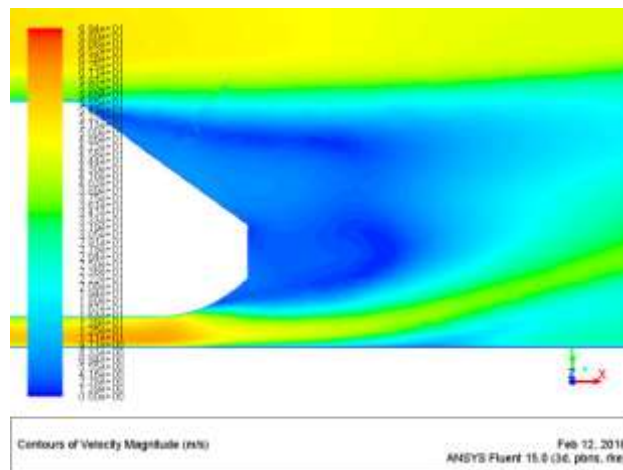**Figure (8):** Pathlines Colored by Velocity Magnitude (m/s)
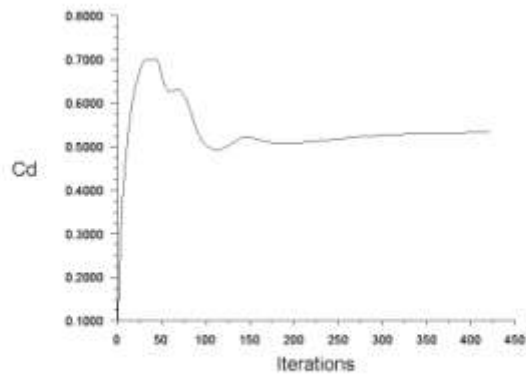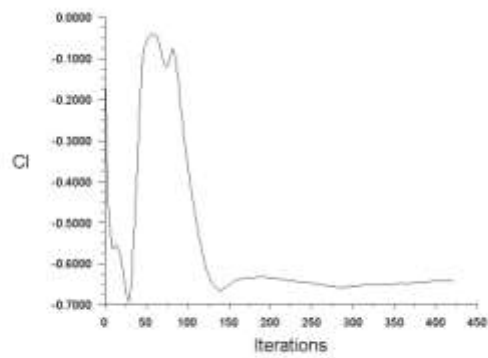

Figure (9): Countours of Velocity Magnitude (m/s)

Figure (10): Air drag coefficient for Hyundai model

Figure (11): Lift coefficient for Hyundai model