 ISBN: 978-81-941253-2-7

AERODYNAMIC FLOW SIMULATION OVER 3D AHMED MODEL USING 3D EXPERIENCE-SIMULIA

Sujit Mishra
Centurion University of Technology and Management Odisha, India
Ashok Misra
Centurion University of Technology and Management Odisha, India

ABSTRACT
The CFD (Computational Fluid Dynamics) analysis using 3D experience- SIMULIA tool used to find the aerodynamic parameter in the automobiles. This analysis helps to understand the savings of fuel consumption, stability of the vehicle and passenger comfort. The airflow over the ground vehicle simulations are analysed and coefficient of drag is calculated. Present study applies 3D Ahmed body (simplified car body) as ground vehicle as the test case. The results presented in the values for drag coefficient obtained along with the contour pressure field and velocity vectors around the body. It is observed that 3D experience SIMULIA tool is one of the convenient simulation approach that delivers accurate CFD simulations results that are matching with wind tunnel experimental results.

Keywords— CFD, 3D Experience- SIMULIA, Aerodynamics, Drag Coefficient, Ahmed Body

INTRODUCTION

Aerodynamics is a branch of fluid mechanics concerned with the study of air flow when interacting with a moving object. It has been playing a key role in the automotive industry in recent years. The development of automotive aerodynamics started with different phases of shape optimization in the early stages of the 1990s, leading to vehicles from the small range to luxury levels. The sedan group is found to be the most fiscal of this wide range of vehicles for the mid-range individuals not only in aesthetics and safety comforts but also for better fuel efficiency.Increased fuel costs and environmental problems are the main challenges of the automotive industry in achieving improved engine efficiency and aerodynamic drag reduction. It could be accomplished either by altering the engine's function or by adding widely used fuel to eco-friendly fuels or modifying the current design of the vehicle. As far as engine optimization is concerned, at the saturation point, we've all achieved the most. Eco-friendly fuels are still a progressing field and it will take a few more years to embrace them globally. Therefore, decreasing aerodynamics drag is the simplest way to increase sedan vehicle efficiency. In this area, studies have been carried out to formulate flow phenomenon techniques over the various sedan shapes, reducing aerodynamic drag & fuel efficiency. Car models carried out studies both by wind tunnels and numerical simulations. If the air moves over the body, as we pass from the front to the rear end, distinct differences occur. In order to visualise the effect of time-average wake structures on the geometry with different configurations at the rear end, Ahmed[1] intended a simplified model. Moreover, Le Good and Garry [2] reviewed the designs of various reference scaled models used in the vehicle production phase in the automotive aerodynamics market.Ahmed, Han, Khan, et.al[3-5] conducted a series of wind tunnel experiments to investigate the pressure and wake structures predicting the difference between the centre and the rear of the vehicle. With the growth and use of CFD packages Bijlani[6] has studied and examined various car models, contrasting the aerodynamic forces acting on them with their effect on fuel consumption and vehicle stability. Some researchers [7, 8, 9] have also adopted various turbulence schemes such as the K-epsilon model, the large eddy simulation and the detached eddy simulation for estimating drag coefficients for the vehicle body. The present study uses the 3D Ahmed Body as the ground vehicle to simulate it in 3D experience- SIMULIA platform through Computational Fluid Dynamics (CFD) framework to obtained aerodynamics features.

METHODOLOGY

A. GEOMETRY MODELLING

The Ahmed model is a simple geometric body that retains the main flow characteristics, particularly the vortex wake flow, where most of the drag is concentrated, and it is a good perfection to be used as a benchmark test. 3D model of Ahmed body consisting of inlet, outlet, nose, upper bottom, slope, back, symmetry. Variation of drag coefficient changes with rear slant angle $35^0$is numerically investigated in turbulent solver schemes designed to meet desirable simulation conditions.