# Aerodynamic flow parameters simulation over a ahmed body using 3d experience- simulia

**Sujit Mishra[1], Sri Devi G [2], Santosh Patro [3], Srinivasa Rao P[4]**

[1]Centurion University of Technology and Management Odisha, India
*sujit.mishra@cutm.ac.in

**Abstract**

Due to the economic developments in society, enormous demand for sedan cars leads to growing competitions in the automotive sectors that undergo numerous tests to improve the fuel efficiency and performance of these cars where aerodynamics plays an avital role. The performance of sedan cars has influenced by aerodynamics due to changes in parameters such as lift and drag forces at high speed. Manufacturers are looking at computational fluid dynamics (CFD) modelling of sedan cars instead of wind tunnel testing with the advancement in computer technology to decrease the testing time as well as the cost of research & development. In the present analysis, a 3D Experience-Simulia Platform simulation has been performed using a Realizable-k-e turbulent scheme to obtain the results of various flow parameters, viz. Over a benchmark test model- 3D Ahmed body, drag force, drag coefficient, turbulent kinetic energy and wake flow structures. It has been observed that in evaluating the aerodynamic parameters, the results of the current simulations satisfy the study requirements.

Keywords: 3D Experience- Simulia, Ahmed Body, Aerodynamics, Realizable k-e model.

## Introduction

Aerodynamics, when dealing with a moving object, is a branch of fluid mechanics concerned with the study of air movement. In recent years, it has played a crucial role in the automotive market. In the early stages of the 1990s, the advancement of automotive aerodynamics began with various phases of shape optimization, leading to vehicles from the small range to luxury levels. For the mid-range people not only in aesthetics and safety comforts but also for better fuel quality, the sedan category is found to be the most fiscal from this large range of vehicles. The big concerns of automotive industries in achieving improved engine performance and aerodynamic drag reduction are increased fuel costs and environmental issues. It could be done either by altering the working of the engine or by supplementing eco-friendly fuels with commonly used fuel or changing the existing nature of the vehicle. As far as engine optimization is concerned, we have all accomplished the most at the saturation stage. Eco-friendly fuels are an environment still under progress and worldwide acceptance will take a few more years. 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


ISSN: 2395-6216