Proceedings of the 25th UKACM Conference on Computational Mechanics
12 –13 April 2017, University of Birmingham
Birmingham, United Kingdom

Simulation of Aerodynamic Behaviour of a Road Vehicle in Turbulent Flow

***Ahmed Al-Saadi, Ali Hassanpour, Yousef Ghaffari Motlagh, Tariq Mahmud**

School of Chemical and Process Engineering, University of Leeds, Woodhouse Lane, Leeds, LS2 9JT

*pmaash@leeds.ac.uk

ABSTRACT

This study concentrates on different aerodynamic drag reduction techniques to reduce the aerodynamic drag coefficient and increase the stability of a three-dimensional full-size road vehicle. There are many modern aerodynamic add-on devices and modifications which are used in this research. All of these aerodynamic devices and modifications are used individually or in combination. Optimization of mesh parameters is carried out by analysis of the mesh data. Unstructured tetrahedral cells are used throughout the global domain to cope with the geometrical complexity of the car model. Inflation layers with prismatic cells are used to provide an accurate estimation of the velocity profiles near the surfaces of the car. Computational Fluid Dynamics (CFD) analysis based on steady state Reynolds-Averaged Navier-Stokes (RANS) turbulence modelling is used. Realizable k–ε, Standard k-ω, Shear Stress Transport k-ω (SST) and a Reynolds Stress Model (RSM) turbulence models are considered in this study. Good agreement has been achieved between the calculated drag coefficient for the baseline models and the experimental data for all types of turbulence models. It is found that the use of some types of aerodynamic modifications and devices can reduce the aerodynamic drag coefficient and increases the car stability.

Keywords: *Simulation; Aerodynamics; Road Vehicle; Turbulent Flow.*

1. Introduction

A rise in fuel prices has led to increasing concern about fuel consumption especially for vehicles. A large part of the engine power is used to overcome the air resistance force and improvement of aerodynamic behaviour can lead to a decrease fuel consumption. Experimental tests and computational simulations have been performed to reduce the drag coefficient of road vehicles and to improve the aerodynamic behaviour. The simple geometry of wagon model was achieved with modifications of the front part by Guo et al. [2] using computational fluid dynamics analysis. The k-ε turbulence model was used to calculate the drag coefficient. The bottom part of the body was assumed as a flat surface. Some parts of car as wheels and rear view mirrors were neglected in modelling to simplify the simulation. This analysis was based on three different slantwise angles of the back windshield. Aljure et al. [1] studied four different LES models, the QR, the VMS, the SIGMA and the WALE, in the bluff bodies using relatively coarse grids. The SIGMA, QR and VMS models were used for the first time to resolve the flow around simplified vehicle models (the Ahmed and the Asmo models were used as baseline models). Three meshes were used with each car model. Number of nodes for the Asmo model was higher than the Ahmed model. Both cases of cars were simulated using the same boundary conditions. The Reynolds number of 7.68×10^5 based on the height of the body was used for both cases. It was found that coarse grids are useful in LES simulations. Khalighi et al. [3] evaluated Immersed Boundary (IB) and body-fitted methods. The IB method does not require mesh to be conformal to geometry and therefore will speed up the grid generation process. The aerodynamic behaviour of Chevy Tahoe 2006 was studied using the Reynolds-Averaged Navier–Stokes solver. Then, velocity, surface pressure and drag coefficient measurements were used to check the simulation results. The drag coefficients for the IB and the body-fitted methods were within 3% and 3–7% of the experimental measurement, respectively.

The drag coefficient of Sports Utility Vehicles (SUVs) is higher than saloon cars because the size and the rear part design of this kind of vehicles. However, the flow field analysis around SUVs is difficult because of the low pressure area behind the car (wake zone). Almost all the previous studies on the aerodynamics of SUVs have focused on the flow characteristics around the car using one turbulence model or one modification of external design [1, 2, and 3]. In this study, four turbulence models and three modifications of external design were used to get all the properties of air around the car and to improve the aerodynamic behaviour.

2. Methodology

The governing equations are solved numerically by finite volume method. The continuity equation and the incompressible Navier-Stokes equation in vector form [5] are given by Eq. (1) and Eq. (2):

$$\nabla \cdot u = 0 \tag{1}$$

$$\frac{\partial u}{\partial t} + (u \cdot \nabla)u = -\frac{1}{\rho}\nabla p + \nu \Delta u \tag{2}$$

where ***u*** is the velocity, t is time, ρ is the density, *p* is the pressure, ν is the kinematic viscosity.

The drag (C_D) coefficient and lift (C_L) are calculated based on the following equations in this study [4]

$$C_D = \frac{F_D}{(\rho V^2 A)/2} \tag{3}$$

$$C_L = \frac{F_L}{(\rho V^2 A)/2} \tag{4}$$

where F_D is the drag force, F_L is the lift force, ρ is the air density, *V* is the inlet air velocity, *A* is the frontal cross sectional area of the vehicle.

3. Numerical Results

Three modifications of external design were used to improve the aerodynamic behaviour of SUVs. Figure 1 shows the baseline model which is used in this study. The dimensions of the baseline model is Land Rover Discovery. Length, height and width without side mirrors of the baseline model are 4835 mm, 1887 mm and 2510 mm, respectively [6]. Unstructured tetrahedral cells are used throughout the global domain to cope with geometrical complexity. ANSYS 16.0 was used to generate mesh with varying levels of refinement. Mesh optimization was carried out by analysis of the mesh data. The CFD simulations are performed with the ANSYS Fluent 16.0 software. The properties of the flow vary a lot behind the car, so most modifications of external design are in the rear part of the car. Four types of turbulence models are used, Realizable k–ε, Standard k-ω, Shear Stress Transport k-ω (SST) and Reynolds Stress Model (RSM), in this study. Table 1 illustrates all the drag coefficients of the baseline model. All Numerical modelling provide good agreement with the experimental measurement [6].

Table 1: The drag coefficients of the baseline model

	Numerical modelling results				Experimental result
	Realizable k–ε	Standard k-ω	SST	RSM	
C_D	0.400112	0.40495	0.41794	0.39701	0.4

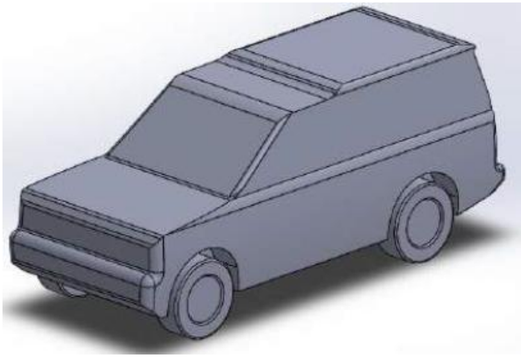

Figure 1: The baseline model of Land Rover Discovery

Spare tyre on the back door of the car, boattail and vortex generators over the end of the roof of the car are used to improve the aerodynamic behaviour of Land Rover Discovery. The Realizable k–ε turbulence model was implemented for the simulation. Figure 2 shows the drag coefficient as a function of Reynolds number for three types of modifications in addition to the baseline model of the SUV. The best modification for the SUV with velocity less than 120 km/h (Re = 1.12×10^7) is a spare tyre because the minimum drag coefficient and low cost. The drag coefficient of the baseline model is 0.4 while 0.372 for the model with spare tyre. Drag coefficient decreases with increasing of Reynolds number except spare tyre which decreases then slightly increases again.

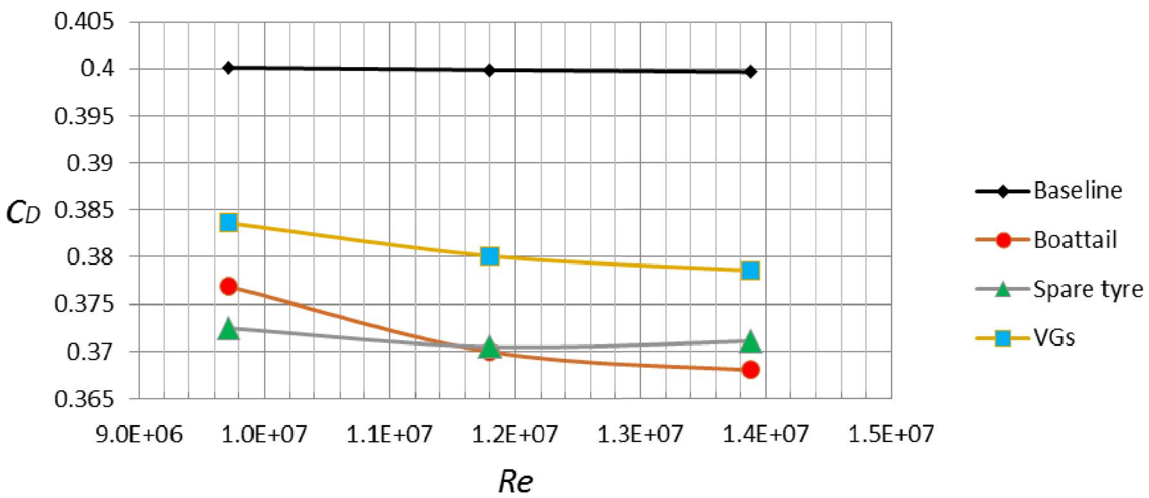

Figure 2: The drag coefficient as a function of Reynolds number

Figure 3 shows the lift coefficient as a function of Reynolds number for three types of modifications in addition to the baseline model of Land Rover Discovery. Vortex generators at the end of the car roof gives higher downforce than other models. The lift coefficient for spare tyre model is independent on Reynolds number as shown in Figure 3. It was found that placing a spare tyre on the back door of the vehicle is the best modifications regarding drag coefficient, lift coefficient and cost.

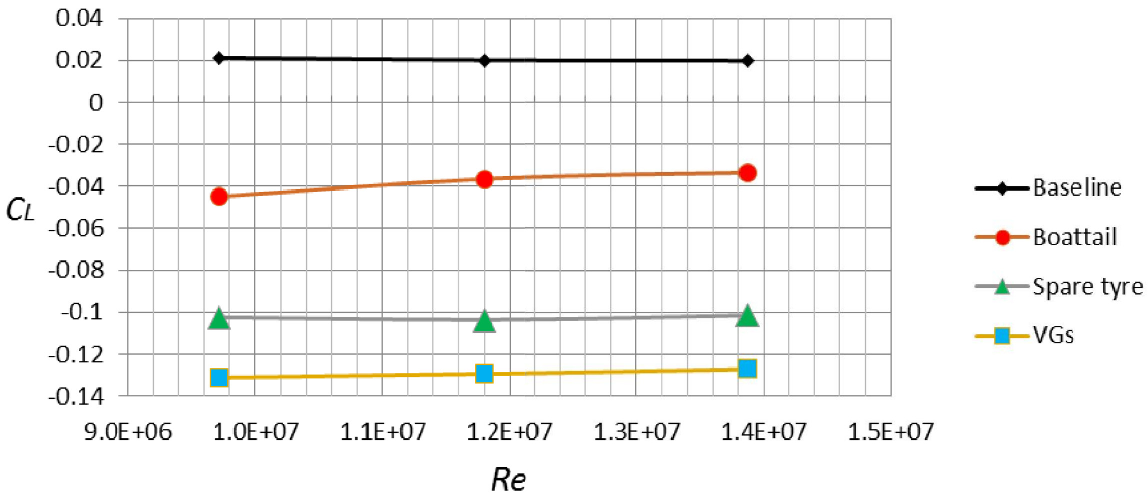

Figure 3: The lift coefficient as a function of Reynolds number

4. Conclusions

Realizable k–ε, Standard k-ω, Shear Stress Transport k-ω (SST) and Reynolds Stress Model (RSM) were utilised to simulate the aerodynamic behaviour of baseline and it was found that all models reasonably predict the experimental measurements. All the modifications to the car provide us with more suitable drag and lift coefficients in comparison with the baseline model. It was found that placing a spare tyre on the back door of the vehicle not only decreases the drag coefficient for a wide range of the Reynolds numbers but also increases the downforce which leads to improvement of aerodynamic behaviour of the vehicle.

Acknowledgements

The first author dedicated his thanks to all who participated in this research, especially the Ministry of Higher Education and Scientific Research (MOHESR) in Iraq and also Al-Qadisiya University in Iraq for sponsoring the first author.

References

[1] Aljure, D. E., Lehmkuhl, O., Rodriguez, I., & Oliva, A. Flow and turbulent structures around simplified car models. *Computers & Fluids* 96, 122-135, 2014.

[2] Guo, L.-X., Y.-M. Zhang and W.-J. Shen. Simulation Analysis of Aerodynamics Characteristics of Different Two-Dimensional Automobile Shapes. *Journal of Computers* 6(5), 999-1005, 2011.

[3] Khalighi, B., S. Jindal and G. Iaccarino. Aerodynamic flow around a sport utility vehicle Computational and experimental investigation. *Journal of Wind Engineering and Industrial Aerodynamics* 107–108: 140-148, 2012.

[4] Obidi, T. Y, *Theory and Applications of Aerodynamics for Ground Vehicles*, SAE International, 2014.

[5] Schuetz, T. C, *Aerodynamics of Road Vehicles*, Fifth Edition, SAE International, 2015.

[6] The official Media Centre for Jaguar Land Rover. [Online]. [Accessed 12 March 2016] 2012. Available from:
http://newsroom.jaguarlandrover.com/en-in/land-rover/press-kits/2011/07/d4_2012my_presskit_060711/