Computational Assessment of Aerodynamic Drag Reduction using Spoilers at the Rear of a Car

Sumit Kanchan
School of Mechanical Engineering, Lovely Professional University, Phagwara-144411, Punjab, India.

Abstract
The motion of vehicle in a fluid exerts a force which affects the performance in terms of fuel efficiency and speed. The drag force acts in the direction opposite of motion which pulls the body back due to the vortexes created in the spaces around it due to the formation of eddies. The nature is attributed to the formation of wake behind the vehicle in motion. To lower the effect Spoilers were introduced to reduce the area of wake formation. This study shows the effect of varying profiles of Spoilers, of varying angle of tilt and the chord length and fixing the position of its location. The result shows that the NACA 2412 reduce the drag force by -3.89N but increases the lift force significantly whereas NACA 2414 series reduce the drag force by -3.41N but the lift force remains almost constant if compared to car without spoilers.

Keywords: Aerodynamic Drag, Spoilers, Computational Assessment

1. Introduction

When a vehicle travels through the fluid, it experiences forces like lift, thrust, drag. The lift creates a low-pressure area above the body so that it could be lifted or fly in air. The thrust or downforce is the multiplication of the body weight acting upon the body which is required for road car or specifically for racing cars as they are meant to be driven fast and the downforce allows them to get proper road grip to accelerate & brake as well. The drag acts as the force which keep on pulling the body back due to the vortexes created in the spaces around it due to the formation of eddies [1]. This phenomena is similar as when a body is immersed or thrown in water as It started to go down it occupies some space due to which the water level rises but, before this when the body starts going down it actually pushing the water down on its face and there is vacancy on its back then to fill the vacancy either the water or fluid will pull the body back or occupies the space by breaking the bondages between them. The larger they cross-section the more eddies it will create & larger drag forces will generate. [4]

The Main Force which opposes the acceleration of the Body is the Aerodynamic drag. In the Field of performance, drag force keeps pulling the car. Whenever the Vehicle burns the Fuel to accelerate and it shows the total of 48-58 % of the fuel energy is wasted to counter the effect [2].

This Aerodynamic drag force, every time it restricts the system to work in proper way due to the formation of eddies or because of the "Separation of Flow" too early, which we have to be delayed. To achieve that either we have to design an Aero-dynamic friendly body or drag free body or make sure that the flowing fluid remain in contact with the bodies profile for as long as possible to delay the Separation of flow [3]. In this work we discussed about spoiler effect on drag reduction. The Lift in the aerodynamic could be defined as the perpendicular constituent to the net force exerting on a body. Drag could be understood as force a flowing fluid applies on a body in the flow direction, like an unwanted result like friction or resistance to motion, which we have to reduce, so decrease of the drag is carefully related with the decrease of fuel ingesting in automobiles. Lift coefficient and drag coefficient were expressed as:

$$C_L = \frac{2F_L}{\rho V^2 A} \quad \text{.....(1)}$$

$$C_D = \frac{2F_D}{\rho V^2 A} \quad \text{.....(2)}$$

where C_L is lift coefficient and C_D is drag coefficient F_L is refer to lift force and it made by a minor pressure differential among the high and lower surfaces of the wing produced by the aerodynamic response to the wing gesture over the atmosphere, F_D is refer drag force, ρ is refer the density and for this case it consider the air density at temperature 25^0 C, V is refer to velocity and this research the velocity be consider 90 km/h until 110 km/h and A is normally refer the frontal area.

The flow round the car is like to that round a rationalize formed object with a spill at the rear. In other words, spill at the rear has the consequence of **postponing flow parting** or shifting the flow separation point upstream or shifting the flow separation point upstream [5]. The objective of the study is for,

Reducing the drag force by optimizing it's the body shape to ensure stream lining or reducing the separation [4].

Performing Comparative drag analysis of sedan with different shapes of indentation with sphere & elliptical volume with different sizes of spoilers is taken and found that drag varies for different Profiles.

Maintaining streamline and delay the flow Separation, for which sharp fillets were avoided, design an aerodynamic friendly body & provide air ducts & enough provisions to the striking air so it transmits minimum resistance to the car motion [1].

Upholding the shape, dropping surface unevenness, less joints of the body or evading shrill fillets, governing lift force for better fuel efficiency and control [6]

The computational studies that are present in this exertion depend on on the RANS calculations and thus, a short overview of the main flow calculations is obtainable in this segment. The incompressible turbulent flow is ruled by the continuity equation and the Navier-Stokes equations that read.

$$\frac{\partial u_i}{\partial x_i} = 0 \qquad (1.1)$$

$$\frac{\partial u_i}{\partial t} + u_j \frac{\partial u_i}{\partial x_j} = -1\frac{1}{\rho}\frac{\partial p}{\partial xi} + \frac{\partial}{\partial x_j}(2\nu s_{ij}) \ , \qquad (1.2)$$

With u_i and p as the prompt velocity and pressure turfs, ρ and ν as the constant density and the kinematic viscosity, respectively [1]

2. Modelling and Simulation: Defining operating conditions

The models based on NACA series were prepared in such a way that one model is prepared based on NACA 2412 and o**n**e other model were prepared based on NACA 2414. The comparison between the drag force and lift force between the three is done later in the study [7]. Fig 3 and Fig 4 show the car with NACA 2412 and NACA 2414 indentation on the top of the car respectively. The CAD model was then imported in the STAR CCM+ and the fluid domain was defined. The specifications of the fluid domain are shown in the table 1. Once the domain is defined it is categorized into physics and mesh continua and the specifications are listed in Table 2 and table 3 respectively.

Fig. 1. Rear View of Modelled car (NACA 2414)

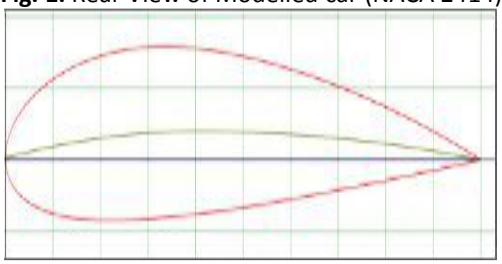

Fig. 2. Sample Profile of NACA 2414

Fig. 3. Front View of Modelled car (NACA 2414)

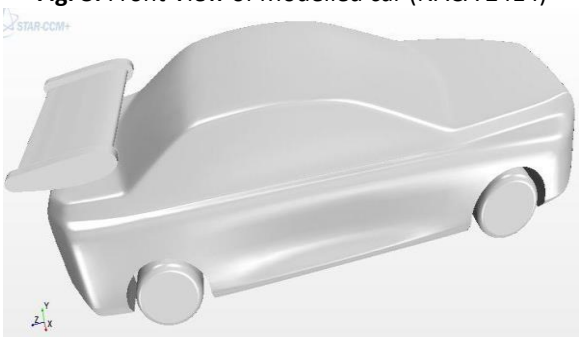

Fig. 4. The NACA series spoiler on the rear of the car.

Defining domain

Table. 1. Domain employed for simulating the geometry.

Face count	12
Coordinate System	Laboratory
Corner 1, m	[-7.8, 0.05, -15.10]
Corner 2, m	[1.6, 6.63, 60.04]

2.1 Continua definition

A. Physics Continua: Two continua were defined for the proper simulation of the geometry formed. The two continua formed were physics continua and mesh Continua. The physics continua models plotted during the process is listed in table 2 and table 3 respectively.

Table. 2. The physics continua defined for simulating the models

Definition	Specification
3D	Space
Implicit unsteady	Time
Gas	Material
Segregated	Flow
Constant density	Equation of state
RANS	Turbulence
K-epsilon turbulence	RANS
Cell quality remediation	Optional models

Mesh Continua: The models employed while defining the mesh conditions were shown in table 3

Table. 3. The mesh continua defined for simulating the models (Sphere and Ellipse

Polyhedral Mesher	Enable Mesh Expansion Control	FALSE
Prism Layer Mesher	Stretching Function	Geometric Progression
Surface Mesher	Do curvature refinement	TRUE
Reference Values		
Base Size	Value	300.0 mm

2.2 Simulation of a car with Spoilers

Fig. 5. shows the profile is indented on the rear of the car and is put to test in under the velocity of 40m/s at the inlet and outlet condition being the atmospheric pressure. The drag force and the lift force are monitored for the values. Fig. 5 shows the car placed inside the fluid domain with NACA 2412

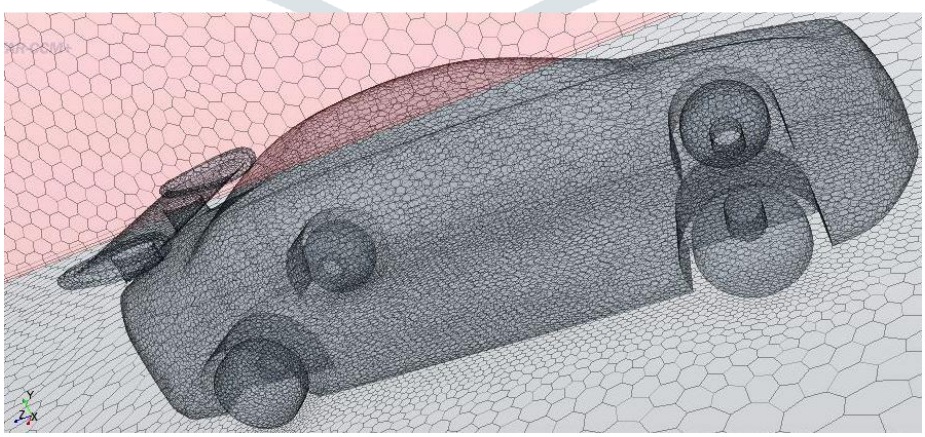

Fig. 5. The front view elliptical shape indented car inside the fluid domain

2.3 Operating Conditions

The specifications of the indentations done on the roof of the car for both types of the Spoilers shape are compared in the table below. The parameters like area, perimeters, AOA and depth were finalized based on the literature survey done.

Type of VG	NO. of VG	Area (in^2)	Perimeter(in)	Dp(mm)
NACA 2414	1	41.85	23.85	8
NACA 2412	1	58.07	26.87	8

Table. 4. The comparison of the two types of Spoiler parameters taken into focus while simulating flow.

3. Results and Discussion

3.1 Result obtained after simulation of car with Spoiler NACA 2414.

The model Car is simulated in STAR CCM+. For the Simulation the new wizard is set up. The SI unit system is adopted and the humidity is introduced in the Fluid selected i.e., Air. The inlet velocity is provided at 40m/s in the Z Direction. The various specifications along with the boundary conditions were as shown in the table 5.

Table. 5. The conditions employed while testing the car with Spoiler NACA 2414.

Coordinate System- XYZ system
Unit system- S.I Unit system
Humidity- 25%
Flow- Laminar and Turbulent
Analysis Type- External
Wall thermal condition: Adiabatic wall
Pressure- 101325 Pa
Temperature- 293.2K
Turbulence intensity: 2% (user defined)

The drag force and the lift force were monitored based on the conditions given. The plots obtained during the test is shown in Fig 9 and Fig 10. The number of cells, faces and edges generated is also summed in the table 6. The result shown in Fig 9 shows the is simulation done for approx. **600 iterations**. The residuals values show the amount of error found with the reference model. The values of drag force obtained in the model employing with Spoiler NACA 2414 is found to be **-3.41N**. This is credited to the boundary layers separation of the flow trajectories over the roof of the car. Thus, the initiatives were taken to delay the boundary layer separation effect by employing a spoiler.

Table. 6. The quantity of cells generated during the flow simulation of car with Spoiler NACA 2414

Volume Mesh	Cells	322026
	Interior Faces	2031357
	Vertices	1707510
Inlet	Faces	653
Outlet	Faces	652
Road	Faces	14578

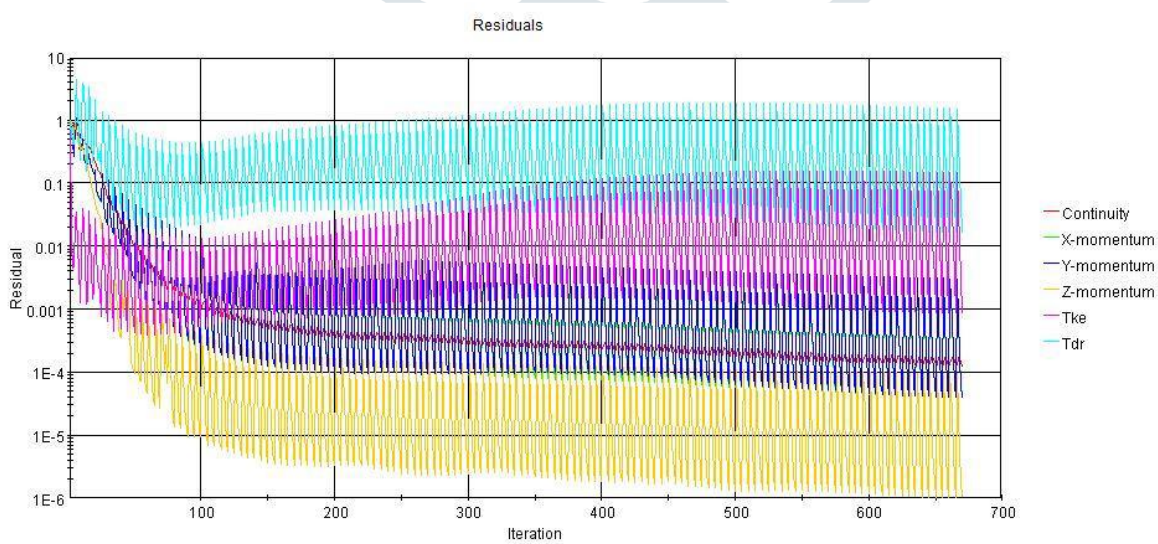

Fig. 9. The result obtained between the drag force and the lift taking into consideration the Prandtl number for car employed car with Spoiler NACA 2414.

3.2 Result Obtained after Simulation of Car with car with Spoiler NACA 2412

The model Car is simulated in STAR CCM+. For the simulation the new wizard is set up. The SI unit system is adopted and the humidity is introduced in the fluid selected i.e., Air. The inlet velocity is provided at 40m/s in the Z Direction. The various specifications along with the boundary conditions were as shown in the table 7.

The drag force and the lift force were monitored based on the conditions given. The plots obtained during the test is shown in the Fig. 10 and Fig. 11. The number of cells, faces and edges generated is also summed in the table 7.

The result shown in Fig 11 shows the simulation done for approx. **550 iterations**. The residuals values show the amount of error found with the reference model. The values of drag force obtained in the model reduce to about **-3.89N.** This is credited to the boundary layers separation of the flow trajectories over the roof of the car. The spoilers delayed the boundary layers separation process and thus resulting the creation of less wake area at the back of the car.

Table. 7. The quantity of cells generated during the flow simulation of car with Spoiler NACA 2412.

Volume Mesh	Cells	672729
	Interior Faces	4517639
	Vertices	3856041
Inlet	Faces	2179
Outlet	Faces	2179
Road	Faces	19640
Units	Preferred System	SI Units

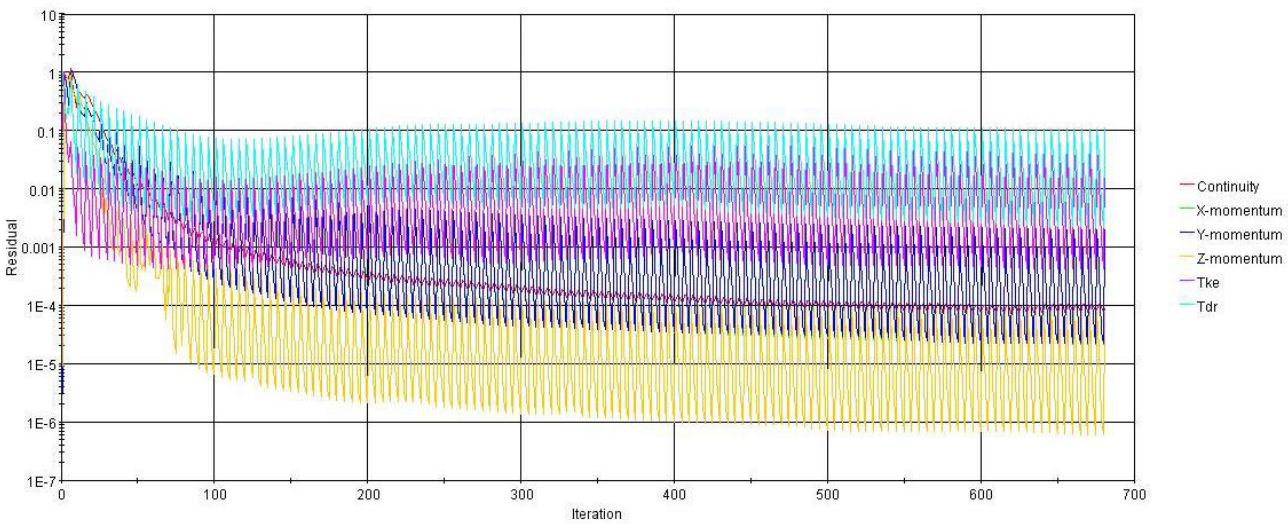

Fig. 10. The result obtained between the drag force and the lift taking into consideration the Prandtl number when car is employed with Spoiler NACA 2412

4. Conclusion

The inferences of this work can be briefed into the succeeding points:

1. While two-dimensional study of aero foil NACA 2412 Wing with Spoiler NACA 2414 in Ground effect considering moving ground frame choosing the right ground height for the front wing geometry, the decision making was crucial because the aerodynamic performance goes high with decrease in drag when the ground proximity increases, but on the contrary the downforce decreases. The prime motive of front wing is to increase the downforce; hence the optimum height of h/c=0.33 has been selected with compromise of drag. For further increment in downforce flap is added near the trailing edge of the main element with the optimized ground height. The double element improves the lift coefficient by ~170%. When the same geometry replicated in the three-dimensional geometry and tested it has been found the aerodynamic performance is decreased by ~300% than the two-dimensional study, the reason could be the surface area and the pressure distribution over the wing span. In similar study of rear wing geometry without considering ground effect it has been demonstrated that the aero foil NACA 2412 gives maximum performance at 14˚AOA and the three-dimensional wing gives ~400% lower aerodynamic performance than the two-dimensional geometry.

2. Spoiler were studied to install immediately upstream of the flow separation point in order to control separation of airflow above the car rear window and improve the aerodynamic characteristics. It was found that the optimum height of the spoiler is almost equivalent to the thickness of the boundary layer (15 to 25 mm). The spoiler is not highly sensitive to these parameters and their optimum value ranges are wide. Better effects are obtained from delta-wing-shaped spoiler than from bump-shaped spoiler.

References

1. Hassan, SM Rakibul, et al. "Numerical Study on Aerodynamic Drag Reduction of Racing Cars." *Procedia Engineering* 90 (2014): 308-313.
2. Sharma, R. B., and Ram Bansal. "CFD simulation for flow over passenger car using tail plates for aerodynamic drag reduction." IOSR Journal of Mechanical and Civil Engineering (IOSR-JMCE) 7.5 (2013): 28-35.
3. Hu, Xu-xia, and T. T. Wong. "A numerical study on rear-spoiler of passenger vehicle." World Academy of Science, Engineering and Technology (2011).
4. Sunanda, A., and M. Siva Nayak. "Analysis of NACA 2412 for Automobile Rear Spoiler Using Composite Material."
5. Kumar, G. Naveen, G. Bharath Reddy, and K. ChandraShekar. "Dynamic Response of NACA 0018 for Car Spoiler using CFRP Material." (2014).
6. Kodali, SHYAM P., and SRINIVAS. Bezavada. "Numerical simulation of air flow over a passenger car and the Influence of rear spoiler using CFD." International Journal of Advanced Transport Phenomena 1.1 (2012): 6-13
7. Milton, S., and S. M. Grove. "Composite Sandwich Panel Manufacturing Concepts for a Lightweight Vehicle Chassis." ACMC, University of Plymouth (1997).