SciEn
Proceedings of 7th International Conference on Mechanical, Industrial and Energy Engineering-Part I
22-24 December, 2022, Khulna, BANGLADESH
SciEn Conference Series: Engineering Vol. 1, 2025, pp 75-80 https://doi.org/10.38032/scse.2025.1.14

CFD Analysis of Car Using NACA 4412 Spoiler

Md. Habibur Rahman Sakib, Mohammad Sultan Mahmud*
Department of Mechanical Engineering, Khulna University of Engineering & Technology, Khulna-9203, BANGLADESH

ABSTRACT
Presently stabilizing the vehicles at high speed with fuel optimization has become a field of concern. These two crucial factors are due to two significant forces drag and lift respectively. CFD is becoming a useful technique in current fluid dynamics research, because to the rapid advancement of digital computers. It brings together fluid mechanics, mathematics, and computer science. This work targets to simulate the drag and lift force on simple SEDAN car modeled in commercial CAD tool using commercial SIMULATION platform. This work focused on variation of dominant forces using an add-on device rear spoiler which was designed using NACA 4412 cross section at different angle of attacks. Then the work focused on evaluating optimal angle of attack at which the spoiler is most economic. The work then focused on variation in flow properties and co-efficient with various Reynolds number. Standard k-epsilon model with standard wall functions and second order upwind equations was used to investigate aerodynamic properties and to acquire the flow structure surrounding the car with rear spoiler. The variation with Reynolds number shows the increase in turbulence and other flow separation phenomenon with increasing Reynolds number. The effect of spoiler with increasing Reynolds number showed that after certain value the lift co efficient becomes almost constant but drag co-efficient increases with increased Reynolds number. With spoiler, it was found that the spoiler was most economic at 8^0 angle of attack by increasing the negative lift twice the previous value without spoiler and drag increment of 10.34% for spoiler with a value of C_D= 0.322 and C_L = -0.27, where without spoiler it stood for C_D= 0.29 and C_L = -0.1325.

Keywords: CFD, Spoiler, Drag, Lift, Downwash, Reynolds Number.

1. Introduction

When a passenger vehicle travels through the air, it is subjected to aerodynamic forces and moments[1]. Flow separation on automobiles usually results in substantial pressure-based drag. It is also capable of generating a lot of lift force[2]. Spoilers are mounted on the rear potion of car to delay the flow separation and remove the unfavorable movement of air around car [3]. Flow-based simulation and analytical models can be used to calculate the effects of air flow over the spoilers. When compared to a theoretical calculation, computational fluid dynamics (CFD) provides a more thorough and precise result. Computational approaches provide more accurate forecasts in less time. It is based on the continuity equation, as well as the energy equation and the momentum equation [4].

Ram & Sharma showed that spoiler with vortex generator has more effectiveness in reducing lift and drag than other add-on devices [5]. Xu Xia Hu and Eric T.T Wong used Gambit 2.0 platform to investigate effects of h/c ratios and shapes of spoiler to reduce drag and lift [3]. Daniel and Norrizal used simulation on Sedan car using ducktail spoiler and rear wing for three various speed in wind tunnel and compared the results [4]. Yuang Cheng showed they effect of spoiler on hatchback cars and investigated c_l for varying pitch angle and found a linear relation [6]. Hamed used canard to reduce the drag of the car [7]. P.G Wright investigated the influence of aerodynamics on the geometry on racing cars [8]. Chen Fu used AKN, SST and RNG k-epsilon model on NASCAR gen 6 cup car to simulate the aerodynamic properties around the car and determined the C_D and C_L for the all three models [9]. M.Palanivendhan investigated the changes in drag due to variations of the angle of attack of vortex generators using STAR CCM+ software [10]. Tomasz Janson and Janusz Piechna used ANSYS-fluent to show the dependency of aerodynamic properties on adjustable aerodynamic parts like flaps size, shape, position and angle of inclination on the aerodynamic lift and drag forces acting on the vehicle [11]. Seong and his teammates investigated the transient spoiler airflow characteristics and the effect of hinge gap and spoiler spoiler location on co-efficient [12]. Ipilakyaa used 20% boundary layer inflation and tetrahedral mesh at 40 m/s velocity and found a lower negative lift with spoiler [13]. Joseph Katz carried out elevated ground plane method to analyze the drag and lift variations on a ¼ scaled race car with two wings and under body vortex generator for achieving the most negative lift [14]. Tien, Zhegai and Zhen analyzed drag co efficient for various Reynolds number and they showed time averaged Pressure distribution for both stationary and rotating wheels. Later they found the separation point to be 225^0 and 220^0 for rotating and stationary wheels respectively [15]. Giacomo Rossitto and his teammates investigated the aerodynamic performance of rounded fastback vehicle using experimental and numerical setup. They compared the effect of blower and spoiler in increasing downward forces [16]. Ram & Sharma used flat plate type spoiler at inclination to reduce lift on sedan car [17]. L.Prabhu and his teammates carried out SOLIDWORKS CFD simulation on sedan car to investigate the effect of rear spoiler in reducing drag and also investigated aerodynamic characteristics around car at five different speeds [18]. Ferrais and his teammates carried out CFD analysis on the baseline configuration of XAM 2.0

*Corresponding Author Email Address: sakib1605007@stud.kuet.ac.bd

vehicle and analyzed the effect of add on devices like spoiler, finlets and front bumpers to see the effect in reducing drag and also showed the contours of pressure distribution around the car [19].

However, this paper is concerned with simple SEDAN car with NACA 4412 spoiler at four different angles to analyze the effect of spoiler with the change in inclination angle that has not been done in above stated reviewed papers. Moreover, NACA 4412 aerofoil is widely spread and most common in aerial vehicles due to its flat bottom. The author tried to reflect its probable potential in using its reversed cross section to manufacture spoiler to be mounted over ground vehicles. This paper also analyzed the effect of forces at different Reynolds number for car without spoiler and with spoiler at fixed angle

2. Mathematical Model:

Despite the fact that the wake behind the vehicle is unstable in the actual flow field at the circumstances at ($Re = 6.5\times10^6$), we solve the governing equations in steady state form because our goal is to get time averaged values of drag and lift coefficients.

The typical k-epsilon turbulence model is used to solve numerically three-dimensional, incompressible, ensemble-averaged, stationary conservation equations of mass and momentum.

RANS equation:

$$\frac{\partial}{\partial x_j}\left(u_j u_i\right) = -\frac{1}{\rho}\frac{\partial P}{\partial x_i} + \frac{\partial}{\partial x_j}\left(2\upsilon S_{ij} - \overline{u_i' u_j'}\right) \quad (1)$$

Mass conservation equation:

$$\frac{\partial u_i}{\partial x_i} = 0 \quad (2)$$

Turbulence kinetic energy (k) equation:

$$\frac{\partial}{\partial x_j}\left(u_j k\right) = -\overline{u_i' u_j'}\frac{\partial u_i}{\partial x_j} - \varepsilon + \frac{\partial}{\partial x_j}\left[\left(\upsilon + \frac{\upsilon_T}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right] \quad (3)$$

Turbulence Dissipation rate ($\mathcal{E}$) equation:

$$\frac{\partial}{\partial x_j}\left(u_j\,\varepsilon\right) = C_{\varepsilon 1}\frac{\varepsilon}{k} = \left(-\overline{u_i' u_j'}\right)\frac{\partial u_i}{\partial x_j} \cdot$$

$$C_{\varepsilon 2}\frac{\varepsilon^2}{k} + \frac{\partial}{\partial x_j}\left[\left(\upsilon + \frac{\upsilon_T}{\sigma_\varepsilon}\right)\frac{\partial k}{\partial x_\varepsilon}\right] \quad (4)$$

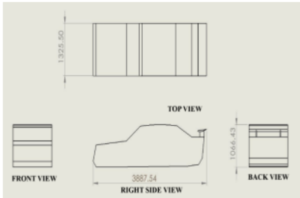
Where, $\upsilon_T = {C_\mu K^2}/{\varepsilon}$

Here,

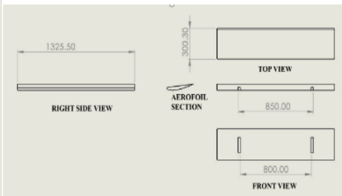
Where υ is the kinematic viscosity of air defined as the dynamic viscosity divided by density, S_{IJ} is the rate of strain tensor defined as $\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_{iJ}}{\partial x_{jI}}\right)/2$ and $u_i' u_j'$ is the Reynolds stress tensor divided by density.

3. Geometric Model:

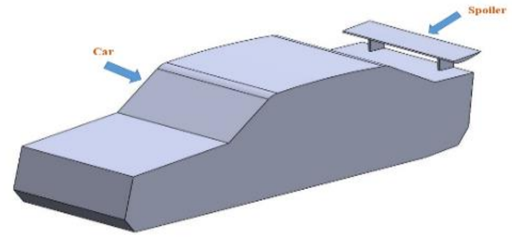
The model of the car having 3887 mm long 1325mm wide and height of 1066 mm is shown in figure 1(a). Spoiler having NACA 4412 reverted airfoil section and 0.3meter chord length was used is shown in figure 1(b). The spoiler was installed at 8^0,10^0,12^0,14^0 angles of attack. Both car and spoiler were modeled using commercial CAD software.



(a)



(b)



(c)

Fig.1 (a) 3D vehicle model with relevant dimensions (meter); (b) Spoiler model with relevant dimensions (meter); (c) CAD MODEL of car with spoiler;

The spoiler of is 300mm height as the chord length for aerofoil section is 0.3 meter and it is located on 262mm from the both side of the rear trunk having total spoiler length of 1325 mm. The supports on which the spoiler is mounted are 800mm apart.

4. Methodology:

4.1 Computational Domain:

The vehicle itself and with spoiler have been installed in virtual wind tunnel box to carry out the simulation. Velocity inlet surface that is in front of car is 0.75 times length of car and pressure outlet on rear portion of car is 1.5 times the length of the car. Because we're more concerned in the rear side of the car, where the "wake of car" phenomena occur, more room has been left in the rear side of the vehicle model to capture the flow behavior primarily behind the car [14].

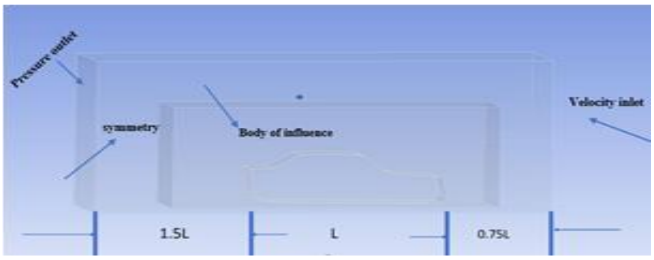


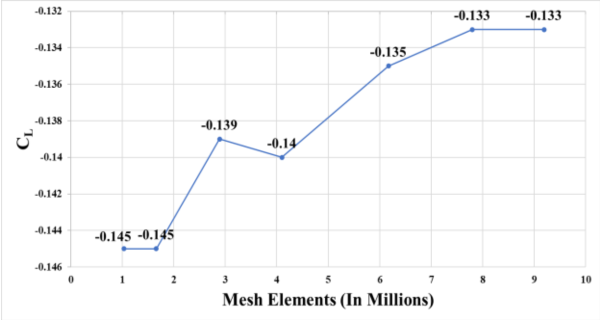
Fig.2 Computitional domain

4.2 Meshing:

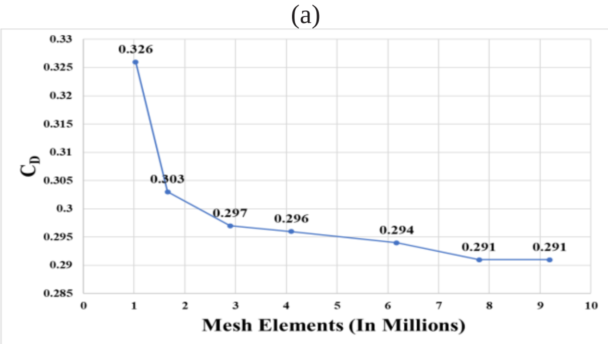
The whole virtual wind tunnel was divided into two regions. Coarser mesh was used in the far region and finer mesh in the body of influence around the car. From the below given graphs it is clear that, the values of drag and lift co-efficient changed with increasing mesh element numbers. So, to get the mesh independent accurate result the values were taken for increasing mesh elements till it did not further depend on mesh elements. It is seen that for 7.8 million and 9.2 millions of mesh the values for lift and drag co-efficient are same at numerical values -0.133 and 0.291 respectively. So, the further works were done for 9.2 million mesh elements.

5. Result & Discussion:

For validation purpose, the work of Ram & Sharma [17] where they found a value of C_D of 0.3512 for the car without spoiler and the author found Deviation of 2%.



(a)

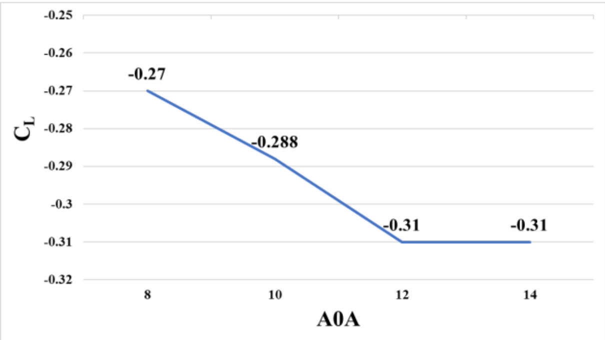


(b)

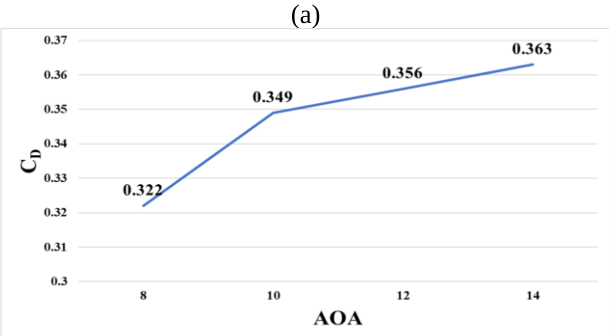
Fig.3 Mesh Independency Test for (a) C_L vs Mesh element number (b) C_D vs Mesh element number

The mesh independency test was done and the mesh conditions were taken where the values of drag and lift co-efficient did not change with further change in size of mesh elements

The result from figure 5(a) & (b) showed that the drag and negative lift both increased with installation of spoiler. The drag and negative lift increased with increase in angle of attack.



(a)



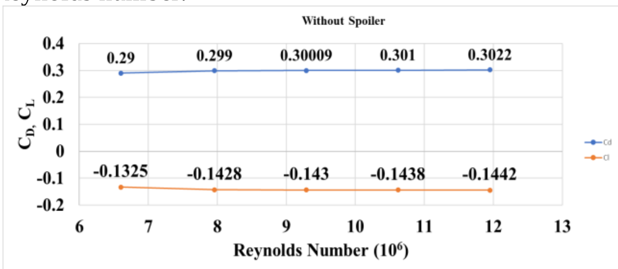
(b)

Fig.4 (a) Variation in C_L with AOA (b) Variation in C_D with AOA

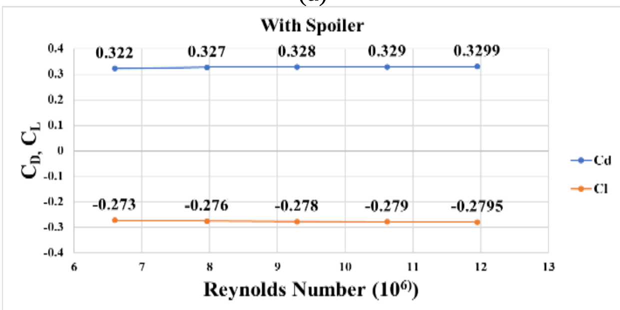
At 8^0 angle of attack, the induced negative lift and drag generated was least with numerical values -0.27 and 0.322, and both the co-efficient increased with the increase in angle of attack of spoiler with lift values -0.288,-0.306,-0.315 and drag values of 0.349,0.356,0.363 at spoiler angle of 10^0 ,12^0, 14^0 respectively.

The negative lift or downwash to drag ratio indicates the amount of negative lift obtained in the Sedan by inducing the drag that is unwanted. And so, the highest negative lift to drag ratio indicates that the negative lift is attained in compensation of less drag induced. The downwash to drag ratio shows the 8-degree angle of attack is more optimum than other angles for having highest downwash to drag ratio with the value of 9.96. The variation in values of co-efficient and flow characteristics were thus visualized for various Reynolds number for NACA 4412 spoiler at 8^0 angle of attack and for car without spoiler.

The figures 6(a)-(b) show the variation in lift and drag co-efficient with the variation in speed thus the corresponding Reynolds number.



(a)



(b)

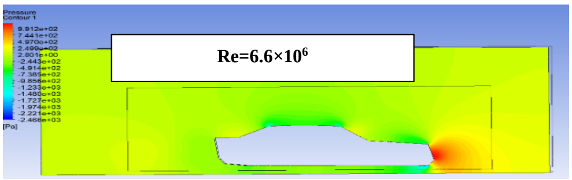
Fig.5 Variation in C_L and C_D with Reynolds Number; (a) Without spoiler, (b) With spoiler

From figure 5(a), the value of C_L and C_D without spoiler at 6.6×10^6 Reynolds number is -0.1325 and 0.29 respectively. That value kept on rising with Reynolds number and at 12.95×10^6 the values were -0.1442 and 0.3022. But the installation of spoiler shows the significant increase in both the values. From figure 5(b) it can be seen that the value of C_L and C_D for car with spoiler at Reynolds number 6.6×10^6 was -0.273 and 0.322, which indicates the increase in stability due to installation of spoiler.

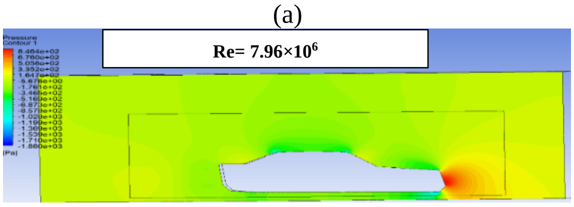
The pressure contour for car with spoiler and without spoiler at various Reynolds number is visualized. Figure 6(a)-(e) and 7(a)-(d) shows pressure contour at various Reynolds number for car with and without spoiler respectively. Figure 6(a) shows that, there is a high pressure region at the front and low pressure region in the bottom side of the vehicle that causes the lift and drag forces to car in motion. Figure 6(b)-(e) also depicts the same pattern but the pressure variations increase with increasing Reynolds number thus the increasing velocity. Figure 7(a) shows that, for mounting spoiler, the high pressure region at the front and low pressure region in the bottom side of the vehicle increased with in comparison with car without spoiler for the similar Reynolds numbers. Moreover, figure 7(b)-(d) shows the pressure contours, that indicate the high pressure region in front of car increases with increasing Reynolds number that show the increasing drag with speed.

Similarly, figure 8(a)-(e) and 9(a)-(e) shows velocity contour at various Reynolds number for car with and without spoiler respectively.

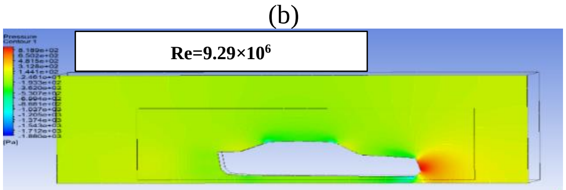
Figure 9(a) shows that, the flow separation phenomena occurs at the rear portion of the car. Figure 9(b)-(e) indicate clearly the flow separates quicker for increasing velocity or the Reynolds number and thus creates turbulence behind the car at movement.



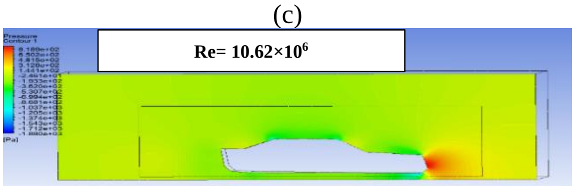
(a)



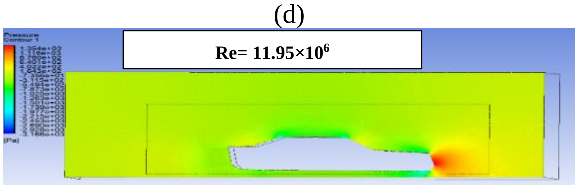
(b)



(c)



(d)

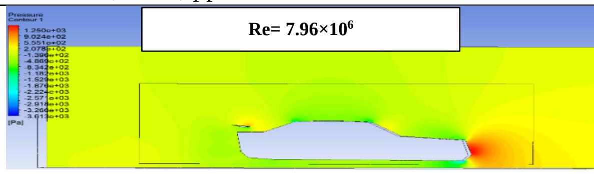


(e)

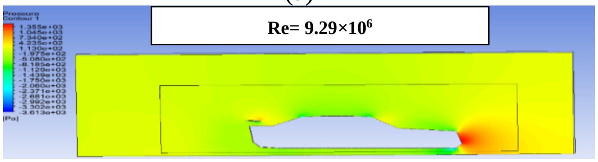
Fig.6 Pressure Contour for various Reynolds Number for car with NACA 4412 spoiler (a) Re= 6.6×10^6 (b) Re= 7.96×10^6 (c) Re= 9.29×10^6 (d) Re= 10.62×10^6 (e) Re= 11.95×10^6.

The visualization for car with spoiler at figure 9(a)-(e) indicate the increment of flow separation span and also depicts the decrement of flow re-circulation behind the car. These two graphical comparisons clearly reflect the increase of stability of car having spoiler at all Reynolds number.

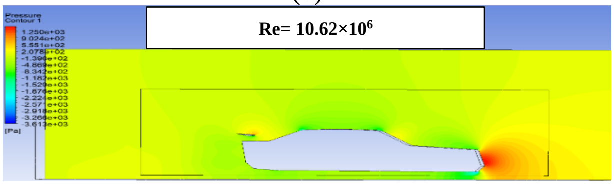
The results show the negative lift increased twice the previous value without spoiler and reduction of lift for using rear spoiler at 8 degree was 103% and the drag increased 10.34% with a value of C_D= 0.322 and C_L = -0.27.



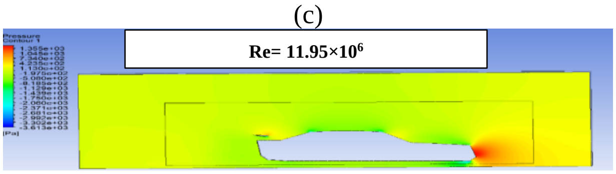
(a)



(b)

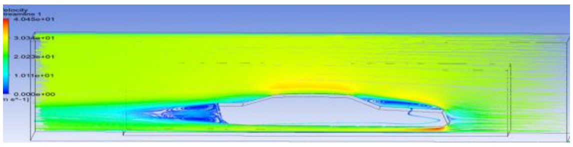


(c)

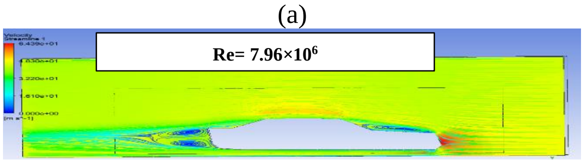


(d)

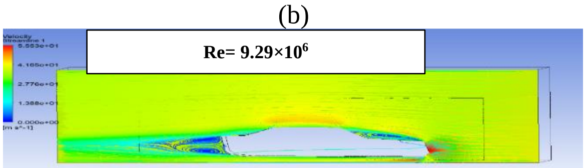
Fig.7 Pressure contour for various pressure contour for car with NACA 4412 spoiler (a) Re=7.96×10^6 (b) Re= 9.29×10^6(c) Re= 10.62×10^6(d) Re= 11.95×10^6.



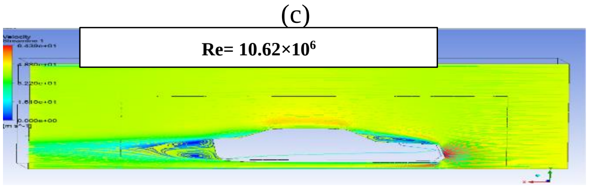
(a)



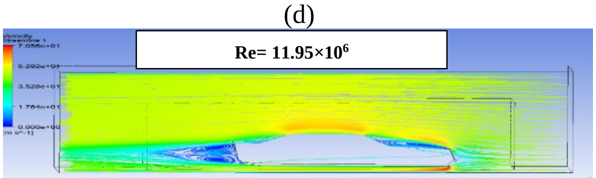
(b)



(c)



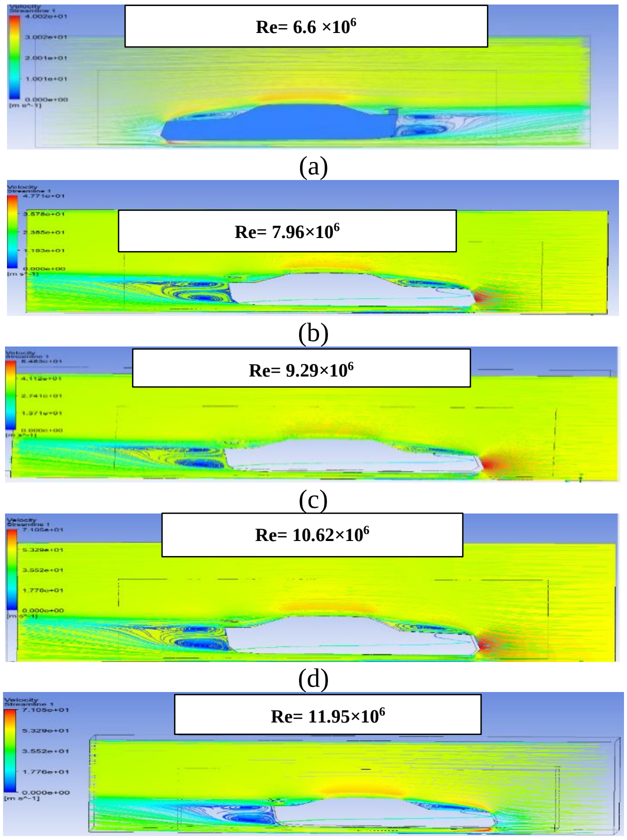
(d)



(e)

Fig.8 (a)-(e) Velocity Streamline for various Reynolds Number for car without spoiler. (a) Re= 6.6×10^6 (b) Re= 7.96×10^6 (c) Re= 9.29×10^6(d) Re= 10.62×10^6(e) Re= 11.95×10^6

The reduction of lift for using rear spoiler at 10 degree was 117.35% and the drag increased 20.34% with C_D= 0.349 and C_L = -0.288. The reduction of lift for using rear spoiler at 12 degree was 130% and the drag increased 22.75% with C_D= 0.356 and C_L = -0.31. The reduction of lift for using rear spoiler at 14 degree was 137.7% and the drag increased 25.17% with drag co-efficient= 0.363 and lift co-efficient= -0.31. This shows that lift to drag difference ratio increases with increase in angle of attack. The variation with Reynolds number shows the increase in turbulence and other flow separation phenomenon with increasing Reynolds number. The installation of spoiler reduces the turbulence on rear side and increases negative lift significantly. The reverted aerofoil shape is reasonable for the downwash whereas, the weight added of the spoiler contributes to the induced drag.



(e)

Fig.9 (a)-(e) Velocity Streamline for various Reynolds Number for car with NACA 4412 spoiler. (a) Re= 6.6×10^6 (b) Re= 7.96×10^6 (c) Re= 9.29×10^6(d) Re= 10.62×10^6(e) Re= 11.95×10^6.

6. Conclusion:

The lift generation of car at motion is proportional with increasing speed. This becomes significant and can be reduced by using spoiler. The spoiler by delaying the flow separation behind car increased downwash by compensation of induced drag. Spoiler were mounted at various angles but here among all the angles of NACA 4412 spoiler 8-degree angle of attack had much significant effect considering both drag and negative lift, where the downwash increased to a value twice of the previous one without spoiler with the reduction in lift of 103% in the compensation of increase in drag of 10.34% for 8-degree angle of attack with value of C_D= 0.322 and C_L = -0.1325 was the most economical and efficient than other angle of attack. The effect of spoiler with increasing Reynolds number showed that after certain value the lift co efficient becomes almost constant but drag co-efficient increases with increased Reynolds number and for that reason the numerical analysis was done till that certain angle of attack as the downwash achievement with least induced drag is our concern. Moreover, the pressure contour and velocity contour for spoiler at 8-degree angle of attack visualized the delaying of flow separation and reduction of turbulence at all Reynolds number with respect to that without spoiler.

References:

[1] I. Kim, X. Geng, and H. Chen, "Development of a rear spoiler of a new type for mini-vans," Int. J. Veh. Des. vol. 48, no. 1–2, pp. 114–131, 2008, doi: 10.1504/ijvd.2008.021155.

[2] Bayindirili, "Numerical drag reduction of a ground vehicle by NACA2415 airfoil structured vortex generator and spoiler" *Int. J. Automotive Technology* vol. 13, no. 2, pp. 293–300, 2012, doi: 10.1007/s12239.

[3] X. X. Hu and E. T. T. Wong, "A numerical study on rear-spoiler of passenger vehicle," *World Acad. Sci. Eng. Technol.*, vol. 81, pp. 636–641, 2011.

[4] D. Syafiq, B. Maji, and N. Mustaffa, "FMC CFD Analysis of Rear-Spoilers Effectiveness on Sedan Vehicle in Compliance with Malaysia National Speed Limit," vol. 3, no. 1, pp. 1–7, 2021.

[5] R. Bansal and R. B. Sharma, "Drag Reduction of Passenger Car Using Add-On Devices," *J. Aerodyn.*, vol. 2014, pp. 1–13, 2014, doi: 10.1155/ 2014/ 678518.

[6] C.S. Yuan, S. Mansor, and M. A. Abdullah, "Effect of spoiler angle on the aerodynamic performance of hatchback model," *Int. J. Appl. Eng. Res.*, vol. 12, no. 22, pp. 12927–12933, 2017.

[7] H. Bagheri-Esfeh, M. Rostamzadeh-Renani, R. Rostamzadeh-Renani, and H. Safihkani, "Multi-objective optimisation of drag and lift coefficients of a car integrated with canards," *Int. J. Comut. Fluid Dyn.*, vol. 34, no. 5, pp. 346–362, 2020, doi: 10.1080/10618562.2020.1766031.

[8] P. G. Wright, "Influence of Aerodynamics on the Design of Formula One Racing Cars.," *Int. J. Veh. Des.*, vol. 3, no. 4, pp. 383–397, 1982.

[9] C. Fu, M. Uddin, and A. C. Robinson, "Turbulence modeling effects on the CFD predictions of flow over a NASCAR Gen 6 racecar," *J. Wind Eng. Ind. Aerodyn.*, vol. 176, no. March, pp. 98–111, 2018, doi: 10.1016/j.jweia.2018.03.016.

[10] M. Palanivendhan, J. Chandradass, P. K. Bannaravuri, J. Philip, and K. Shubham, "Aerodynamic simulation of optimized vortex generators and rear spoiler for performance vehicles," *Mater. Today Proc.*, vol. 45, pp. 7228–7238, 2021, doi: 10.1016/j.matpr.2021.02.537

[11] A. Mechanics, "Numerical Analysis of Aerodynamic Characteristics of a of HighSpeed Car," 2015.

[12] S. W. Choi, K. S. Chang, and H. Ok, "Parametric study of a rapidly deploying spoiler with two-equation turbulence models," 39th AIAA Aerospace Science Meeting & Exhibit, doi: 10.2514/6.2001-864 T. D. Ipilakyaa, L. T. Tuleun, and M. O. Kekung, "Computational fluid dynamics modelling of an aerodynamic rear spoiler on cars," *Niger. J. Technol.*, vol. 37, no. 4, p. 975, 2018, doi: 10.4314/njt.v37i4.17.

[13] J. Katz and D. Garcia, "Aerodynamic effects of Indy car components," *SAE Tech. Pap.*, vol. 111, pp. 2322–2330, 2002, doi: 10.4271/2002-01-3311.

[14] T. P. D. Z. G. and Z. Chen, "Numerical simulation of flow field around the race car in case Stationary wheel and rotating wheels," *Int. J. Numer. Methods Heat Fluid Flow*, vol. 24, no. 4, 2014, doi: 10.1108/HFF-02-2014-0034.

[15] G. Rossitto, C. Sicot, V. Ferrand, J. Borée, and F. Harambat, "Aerodynamic performances of rounded fastback vehicle," *Proc. Inst. Mech. Eng. Part D J. Automob. Eng.*, vol. 231, no. 9, pp. 1211–1221, 2017, doi: 10.1177/0954407016681684.

[16] R. Sharma and R. Bansal, "Drag and Lift Reduction on Passenger Car with Rear Spoiler," *Res. Dev.*, vol. 3, no. 3, pp. 13–22, 2013.

[17] Prabhu, L. & Krishnamoorthi, Sangeetha & Gokul, P & Sushan, Nandhu & Harshed, P & Jose, Aviss. (2020).Aerodynamics analysis of the car using Solidworks flow simulation with rear spoiler using CFD. IOP Conference Series: Materials Science and Engineering. 993. 012002. 10.1088/1757-899X/993/1/012002.

[18] M. AL-Rawi and A. Oumssount, "One-Way Fluid Structure Interaction of a Go-Kart Spoiler Using CFD Analysis," The 13th Conference of the International Sports Engineering Association, Jun. 2020, doi: 10.3390/proceedings2020049051.

[19] Paul, A.R., Jain, A. & Alam, F. Drag Reduction of a Passenger Car Using Flow Control Techniques. Int.J Automot. Technol. 20, 397–410 (2019). https://doi.org/10.1007/s12239-019-0039-2.

NOMENCLATURE

C_P :Co-efficient of pressure
C_L :Co-efficient of lift
C_D :Co-efficient of drag
ρ :Air density, kgm^{-3}
k :Turbulent kinetic energy, m^2s^{-2}
ε :Rate of dissipation of turbulent kinetic energy, m^2s^{-3}
Re :Reynolds Number