International Journal of Mathematics and Physical Sciences Research ISSN 2348-5736 (Online)
Vol. 10, Issue 1, pp: (38-44), Month: April 2022 - September 2022, Available at: www.researchpublish.com

Using CFD Analysis to Investigate the Appropriate Height of the Rear Spoiler on a Car

Mahmoud Ibrahim Youssef

Department of Automotive and Tractors Engineering, Faculty of Engineering, Helwan University

DOI: https://doi.org/10.5281/zenodo.7036018

Published Date: 30-August-2022

Abstract: **This work proposes an efficient numerical model based on the Computational Fluid Dynamics (CFD) approach to get the flow structure around a car with a distinct height of Spoiler. The main target of the project is to show such aspects using a CFD packages. Our project is to investigate the aerodynamics characteristics of a car with rear spoiler and without rear spoiler and investigate the suitable height of the rear spoiler. Four totally different velocities are chosen for this analysis. It's found that the installation of a spoiler at the height of 371mm upper surface of the vehicle trunk. At this position, the total drag coefficient reduction of 26%**

Keywords: **CFD, spoiler, downforce, drag, vehicle trunk, drag reduction.**

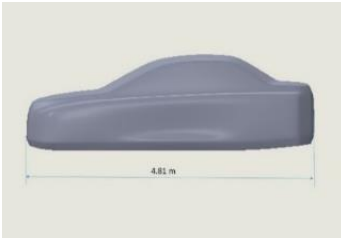
1. INTRODUCTION

Nowadays, vehicles are modified to form it sportier. Therefore the makers are trying to find new ways and discovering new technologies and procedures to reduce fuel consumption and improve vehicle efficiency. Many strategies are accustomed reduce aerodynamic drag for controlling the flow separation at the rear end. These technique is hanging add-on device to reduce the aerodynamic drag and improvement the stability of the vehicle by increasing the downforce on the vehicle trunk. Once the car is driven in high speed condition, the car has high orientation to raise over. This is possible to happen because of high pressure in the air in the front wind shield it inflicting the pressure to drop. Then the lower pressure lifts on the car roof because the air passes over it. Drag forces is determined by the airflow over a vehicle, which in turn affects the car's performance and design of efficiency. Testing instrumentality has been done to measure both the vertical and horizontal parts of air resistance on a model car. Sport cars are affected by the drag force because it works opposite to the direction of movement. About 30% to 40% of the total fuel energy is lost to overcome road resistance, about 10% to 20% for operating electrical appliances and 50% to 60% to overcome the drag force. Thus reducing of aerodynamics drag has become the firstconcern in vehicle aerodynamic great efforts in researches and analyses are utilized for better fuel economy and performance of road vehicles. Most of the sedan cars were manufactured in twentieth century. From highest speed Hennessey venom GT reaching up to 270.49 mph, Bugatti to the luxurious Rolls Royce phantom and far lot of sport cars. Depending on the customer's alternative, personal cars starting from hatch back, sedan and SUV have seen to major changes in their style of shape. It will be provide more stability, better performance to increase the comfort of the vehicle.

Problem description

In this study, a generic model, a reference vehicle model introduced by Cakir 2012, is adopted for simulating the sedan-vehicle. The vehicle fitted with a spoiler at totally different height at a similar angle of attack. Study drag force and lift force of the spoiler at totally different heights by performing fluent tool (CFD). 3D modelling software SOLIDWORK is used for designing and ANSYS CFD is used for both meshing and solver.

a) Front view



b) Side view

Dimensions view

Figure 1. Geometric model of the vehicle

Four models for variedspoiler mounting heights are showing in figure 2. Just one rear wing is taken into account at the time in every simulation.

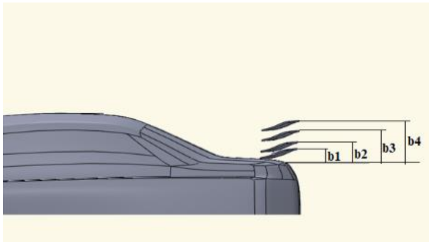


Figure 2. The model with four different mounting heights of the rear wing

II. COMPUTATIONAL DOMAIN AND INLET BOUNDARY CONDITIONS

As the aim of this work is to reproduce computationally the wind tunnel flow over road vehicles, the computational domain was adjusted to the main dimensions of structure test section. In attempting to reproduced it. The width is 4L, and therefore the height is 2L, and 8L long, see figure 3.

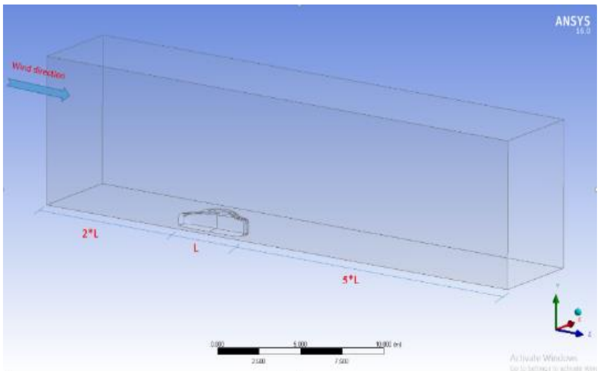


Figure 3. Enclosure Isometric View

The boundary condition at the inlet of the domain requires a totally developed flow with boundary layers on roof, walls and floor, as this is the case in the wind tunnel test section.

III. MESHING

ANSYS Fluent Meshing has generated the meshes with the sizing parameters. Table 1 presents the mesh parameters and figure 6 shows the mesh property.

Table 1: The global mesh sizing

Global mesh sizing setting	
Use Adv. Size Fun.	On Proximity and Curvature
Relevance centre	coarse
Curvature Normal Angle	12
Minimum Size	17.823 mm
Maximum Size	500 mm
Growth Rate	1.2 (20%)
Inflation	
Use Automatic Inflation	Program Controlled
Inflation Option	First Aspect Ratio

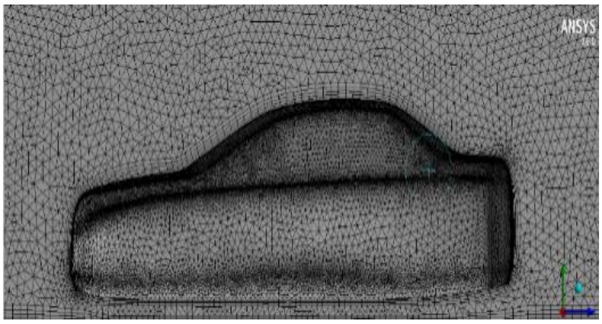


Figure 4. The Mesh Sizing

To improve the mesh quality round the car model, the virtual car-box was generated as shown in figure 5.

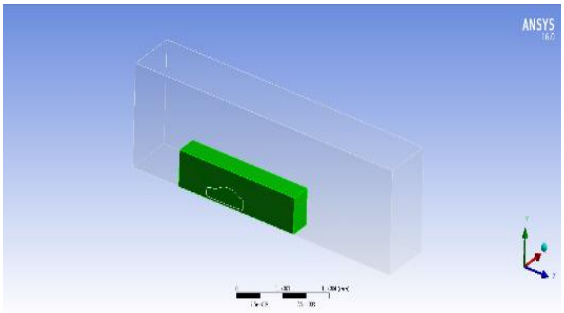


Figure 5. The virtual vehicle-box orientation

The final meshing is give in figure 6. The similar procedure to form high resolution meshing has been followed for all cases.

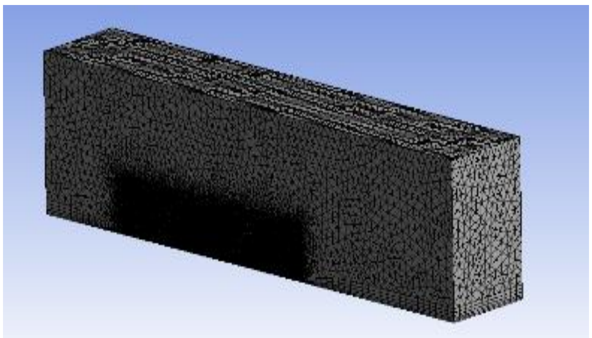


Figure 6. The Final Mesh

IV. SOLVING AND VALIDATION

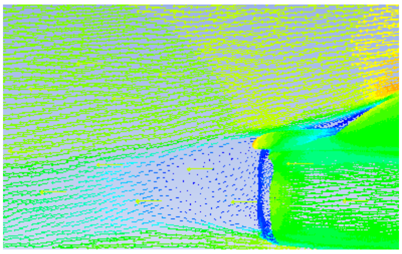
The final step is ANSYS Fluent Setup. The solver settings and boundary conditions for all cases are shown within the Table 2.

Table 2. The solver setting

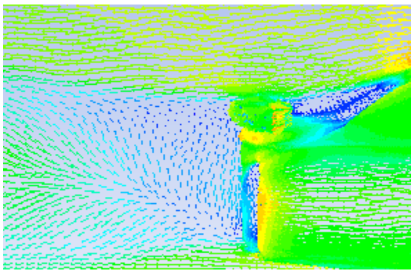
Magnitude and Direction (m/s)	30.6	36.1	41.7	42.7
Gauge Pressure Direction	Normal to Boundary			
Backflow Turbulence Intensity	10%			
Fluid type	Air			
Density	$\rho = 1.175$ kg / m^3			
Kinematic viscosity	$v = 1.8247 \times 10^{-5}$ kg / m.s			
Turbulent model	k-ε (2-eqn)			
k-ε Model	Realizable			
Near-wall Treatment	Non-Equilibrium Wall Functions			
Scheme	Coupled			
Gradients	Least Squares Cell Based			
Iteration	First Order Upwind for the first 100 iterations, Second Order Upwind until converged			
Flow Courant Number	50			
Under-Relaxation Factors	0.8 for the first 100 iterations, then 0.95			

V. RESULTS

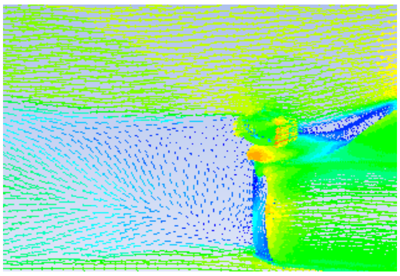
By comparison velocity magnitude (choosing the highest speed of 42.7 m/s) of with/without spoiler heights, (figure 7) it's seen that the recirculation zone behind the rear end of car is totally different in all cases.



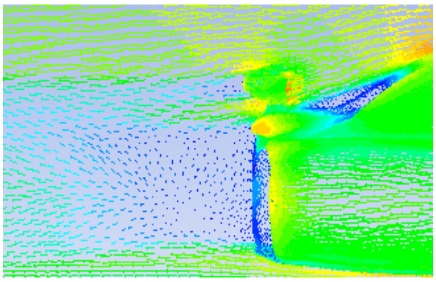
Car only



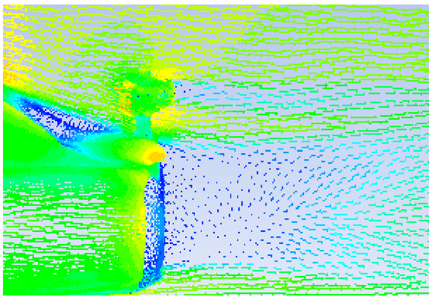
At b1= 76 mm



At b2= 170 mm



At b3= 273mm



At b4= 371 mm

Figure 7. The velocity magnitude for all cases at 47.2 m/s

Five graphs were plotted in figure 8. From the graphs, drag force was found directly proportional to the squared speed in all cases. This indicates that when speed was increased, the drag force also increased. Therefore, the results detected using five component balances for drag force was accepted.

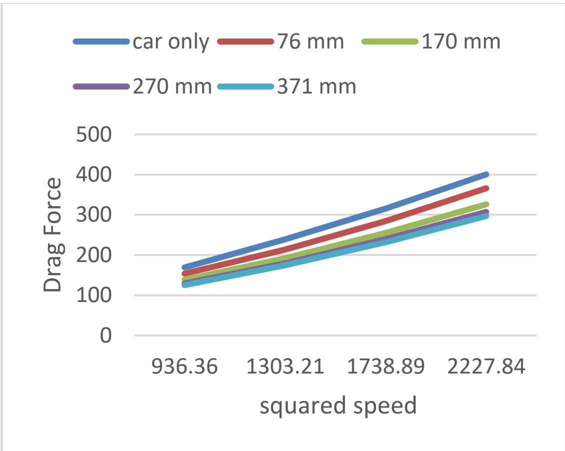


Figure 8: Graphs of Drag Force Vs squared speed

The graphs in figure 8 was plotted according to equation:

$$F_D = \frac{1}{2}\rho V^2 A C_d$$

This equation was used to validate the analysis results, where the increased squared speed would also give the effect increasing drag force. The axial force that hit the frontal area of the scale car also would increase and result in higher drag force.

Figure 9 shows that the value of drag coefficient decreased when the speed increased. This graph proves that spoiler is an aid that helps to reduce the value of drag coefficient reduced until reaching a constant value at the end of the speed of 47.2 m/s.

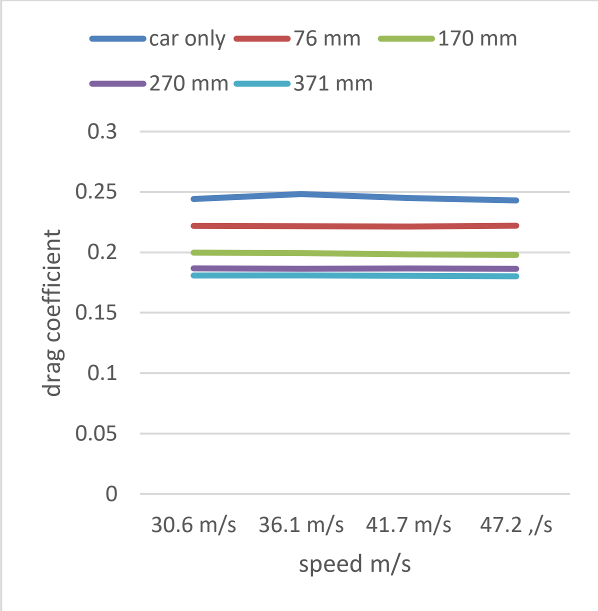


Figure 9: Graph of drag coefficient vs. speed

Figure 10 was plotted to see the difference in drag coefficient value. This graph shows that when spoiler height increased, the drag coefficient decreased until it reached the optimum height at 371 mm.

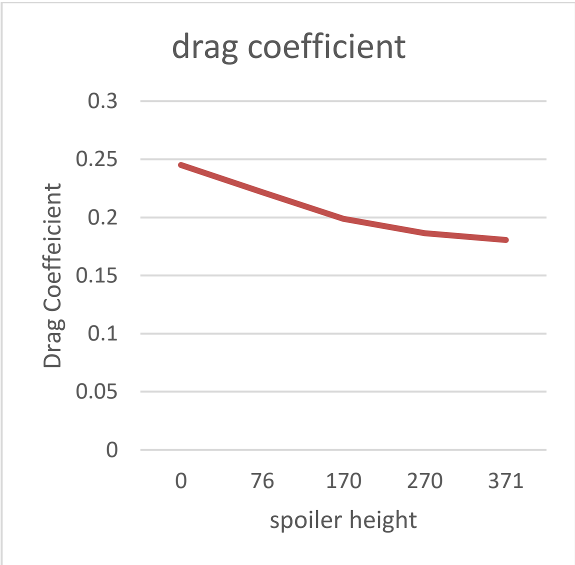


Figure 10: graph of drag coefficient vs. spoiler height

Based on the value of drag coefficient when spoiler was used, the percentage of drag coefficient reduction was calculated. Then, comparison was made to get the most suitable height that should be used for the model. Table 3 shows the percentage of drag reduction for all spoiler heights.

Table 3. Drag reduction

Spoiler heights, h (mm)	% C_d Reduction
76	9.53
170	18.89
270	23.89
371	26.32

VI. CONCLUSIONS

Effect of the spoiler mounting height on the vehicle is computationally studied by using k-epsilon (2 eqn) model with realizable and non-equilibrium wall function. All analysis has been carried out using ANSYS CFD Fluent for the model with four different heights of a wing. As a conclusion, the value of drag coefficient for the car model when the spoiler is used is 0.180588 at the height of 371 mm, with the total drag coefficient reduction of 26.32%. Analysis has been carried out using ANSYS CFD fluent for the model with four different heights of spoiler, and the results have been compared with the model without spoiler to find the height that gives the most reduction in drag coefficient value.

REFERENCES

[1] John J. Bertin, "Aerodynamics for Engineers", Prentice Hall; 5th edition, New Jersey, June 2008

[2] Jiyuan Tu, Guan Heng Yeoh and Chaoqun Liu, "Computational Fluid Dynamics: A Practical Approach", Butterworth-Heinemann; 1st edition, Burlington, MA, November 2007

[3] Oleg Zikanov, "Essential Computational Fluid Dynamics", John Wiley & Sons, Inc. Hoboken, New Jersey, March 2010

[4] C. H. K. Williamson, "Three Dimensional Vortex Dynamics in Bluff Body Wakes", Experimental Thermal and Fluid Science, Volume 12, February 1996, p. 150-168

[5] Wolf-Heinrich Hucho, "Aerodynamics of Road Vehicles: From Fluid Mechanics to Vehicle Engineering", Society of Automotive Engineers Inc; 4th edition, Warren dale, Pa, February 1998

[6] Marco Lanfrit, "Best practice guidelines for handling Automotive External Aerodynamics with FLUENT", Fluent Deutschland GmbH, 64295 Darmstadt/Germany, February 2005

[7] W. Seibert "CFD in Aerodynamic Design Process of Road and Race Cars", Fluent Deutschland GmbH, FLUENT Technical Notes TN155, Presented at European Automotive Congress, Bratislava, Slovakia, June 18-20 2001

[8] Klaus Gerstein, E. Krause, H. Jr. Oertel, C. Mayes "Boundary-Layer Theory", Herrmann Schlichting, 8th Edition, International Journal of Pure and Applied Mathematics Special Issue 14591

[9] ANSYS Fluent 12.0 Theory Guide, [Online]. - July 26, 2011.-http://www.cadfamily.com/downinfo/285585.html.

[10] ANSYS FLUENT Theory Guide [Book] / auth. ANSYS Inc.. - U S A : ansys.com, 2017.

[11] CFD Study on Aerodynamic Effects of a Rear Wing/Spoiler on a Passenger Vehicle [Book] / auth. Cakir Mustafa. - [s.l.]: Mechanical Engineering Master Theses, 2012.

[12] CFD Analysis on the Aerodynamic Effects of Spoiler at Different Angle on Car Body [Journal] / auth. Akhilesh Singh Tomar [et al.]. - [s.l.]: International Journal of Innovative Technology and Exploring Engineering, 2019. - 7: Vol. 8.