INTERNATIONAL JOURNAL OF RESEARCH IN AERONAUTICAL AND MECHANICAL ENGINEERING

ISSN (ONLINE): 2321-3051

Vol.4 Issue 9, September 2016 Pgs: 1-8

Experimental and Numerical Investigation of Airfoil Shaped Spoiler

Pon Maa Kishan A

Research Scholar, ponmaakishan@gmail.com

Corresponding Author: Department of Mechanical Engineering, Regional Campus, Anna University Tirunelveli, India-627007. E-mail: - ponmaakishan@gmail.com

Abstract

A vast amount of research has been going on to optimize and select the best shaped spoiler configurations for the purpose of achieving minimum drag and minimum lift. Airfoil shaped spoiler used in conjunction with plain airfoils for the purpose of reducing induced drag, vortices, and decreasing lift. Spoiler also increases the traction of the vehicle. CFD analysis was carried out on NACA 2412 airfoils shaped spoiler of chord 6 inch and span 12 inches. The CFD values of drag and lift coefficients achieved were found to be in good agreement with the experimental data thus, the CFD analysis was then extended to analyze NACA 2412 airfoils of chord 6 inch and span 12 inches. The thickness of winglet was half of the airfoil chord length. Spalart-Allmaras model is used for the purpose of CFD analysis, coefficient of lift (C_L), coefficient of drag (C_D), lift/drag (L/D) ratio were determined for these spoiler at various angle of attacks and their aerodynamic characteristics were compared to other NACA 4 digit airfoils.

Keywords: NACA, Airfoils, Spoiler, CFD.

1. Introduction

The history of Automobile spoiler was way back from the 1960's when race cars still looked more or less like cars on the normal street. Spoilers were actually put to use by NASCAR in racing teams to create a downward force on the rear side. The spoiler was first popularised by sports and racing cars in the 1970's and today this add-on can be spotted on all types of cars.[1] The main function of a spoiler is diffusing the airflow passing over and around a moving vehicle as it passes over other vehicles. This diffusion is accomplished by increasing amounts of turbulence flowing over the shape, 'Spoiling' the turbulent flow and providing a cushion for the laminar boundary layer often spoilers are added solely for the appearance with no knowledge towards practical purpose. Spoiler is to allow for better airflow over and around the car, which in turn creates a better grip or traction to vehicle on the road.

1.1 Problem Statement

When a driver drives his or her vehicle in high speed condition, especially at highway which is speed limit 110 km/h, the car has high tendency to lift over and also drag will be produced at high range. This is possible to happen because as the higher pressure air strike in front of windshield and travels over the windshield; it accelerates, causing the pressure to drop. To reduce lift and drag that acted on the rear trunk, rear spoiler can attach on it to create more high pressure. Spoilers are primarily used in sedan type car. This act like barriers to air flow in order to build up higher air pressure in front of the spoiler.

1.2 Drag

Drag sometimes usually called air resistance, a type of friction, or fluid resistance, another type of friction or fluid friction is force acting opposite to the relative motion of any object moving with respect to a surrounding fluid.

The following are the basic types of drag

1. Pressure drag
2. Friction drag
3. Induced drag

1.3 Drag Coefficient

In aerodynamics, the drag coefficient is one of the dimensionless quantities that are used to measure the drag or air resistance of an object or body in a fluid environment, such as water and air. C_D is used in the drag equation, where a lower drag coefficient always indicates that the object will have less aerodynamic drag. Drag coefficient is always associated only with a particular surface area.

1.4 Bernoulli's principle

One of the important principles in the field of aerodynamics is Daniel Bernoulli's principle. It is the principle which says that when there is an increase in flow velocity of a gas or fluid there is a decrease in pressure in a fixed volume. This is due to Newton's concept, as he said that energy cannot be created neither destroyed. This theory is used very extensively in aeronautical applications. The shape of an aircraft wing only causes the air to flow faster over the top surface than the bottom one. The Bernoulli's principle says there is a lower pressure on the top surface compared to the bottom surface and so that this creates lift.

1.5 NACA 2412 airfoil

The NACA airfoils are developed by the National Advisory Committee for Aeronautics (NACA). The shape of the NACA airfoils is described only using a series of digits following the word "NACA". The parameters in the numerical code can be entered into the equations to precisely generate the cross-section of the airfoil and calculate its properties. [2] The NACA four-digit wing sections which define the profile by For example, the NACA 2412 airfoil has a maximum camber of 2% located 40% (0.4 chords) from leading edge with a maximum thickness of 12% of the chord. Four-digit series airfoils by default it will have maximum thickness at 30% of the chord (0.3 of chord) from the leading edge. The NACA 0015 airfoil is symmetrical; the digit 00 indicating that it has no camber. The 15 indicates that this airfoil has a 15% thickness to chord length ratio, it was 15% as thick as it is long. Four- and five-digit series airfoils can be modified with a two-digit code preceded by a hyphen as the following sequence:

1. One digit describing the roundness of the leading edge with digit 0 being sharp and 6 being the same as well as the original airfoil, and larger values indicating a more rounded leading edge.

2. One digit will be describing the distance of maximum thickness starts from the leading edge in tens of percentage of the chord.

For an example, the NACA 2234-05 is a NACA 2234 airfoil with a sharp leading edge and the maximum thickness 50% of the chord (0.5 of chord) from the leading edge. In addition, for a more precise description of airfoils all numbers can be only presented as decimals.

2. Methodology

There are many type of spoiler out there and the design must be capable and suitable with car model. The designs then build up in CAD software and analyze in CFD software. The result of analysis will be either use or refine if needed. Figure 1 shows the analytical methodology of airfoil shaped spoiler. The result of analysis will be analyze and discuss for the next step and summary will be done to conclude the project. May be some designs are not suit with the model.

The test car model is sedan type. So, rear spoiler to attach at the rear boot must suit and fix with that kind of car design. Spoiler like deck lid or free-standing airfoil must be used, which is suitable with sedan type car.

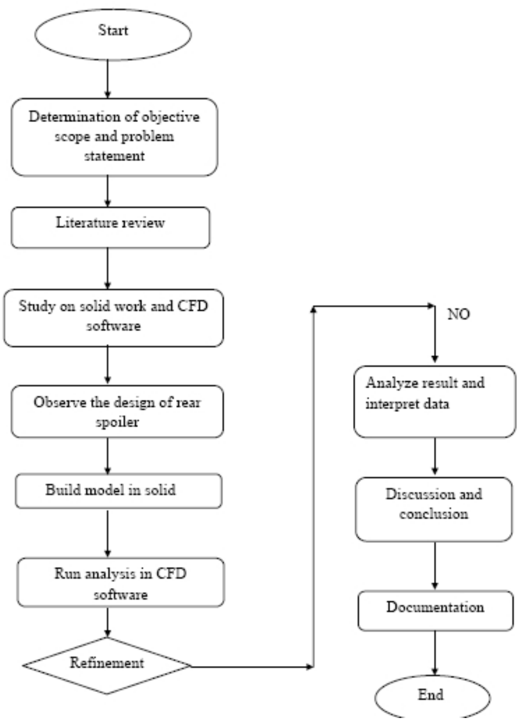

Figure 1: Methodology

3. Design Tool - Solidworks

The parametric tool of Solidworks means that the dimensions and relations drive the geometry, not the other way around. The dimensions in the airfoil sketch can be controlled independently, or by relationships to other parameters inside or outside of the sketch. Building a model in Solid Works usually starts with a 2D sketch (although 3D sketches were available through part design). The sketch consists of geometry such as points, lines, rectangles, arcs, conics (except the hyperbola), and splines. Dimensions which are required are added to the sketch to define the size and the location of the geometry. Figure 2 shows the final part diagram of spoiler and it is converted to .igs format which is required to import the geometry in ansys fluent.

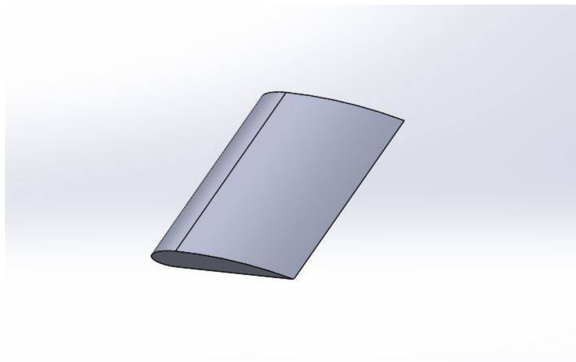

Figure 2: 3D Part design of Spoiler (All Dimensions are in cm)

4. Analysis Tool – Ansys Fluent

With different directional CAD connectivity, powerful highly-automated meshing tool, a project-level update auto mechanism, pervasive parameteric management and integrated shape optimization tools, the Ansys Workbench platform delivers unprecedented productivity, enabling Simulation-Driven Product Development.

Step 1

Then designed geometry of spoiler is imported in ansys fluent, figure 3 shows the airfoil shaped spoiler is imported and boundary is set using fully closed box type enclosure under uniform flow conditions.

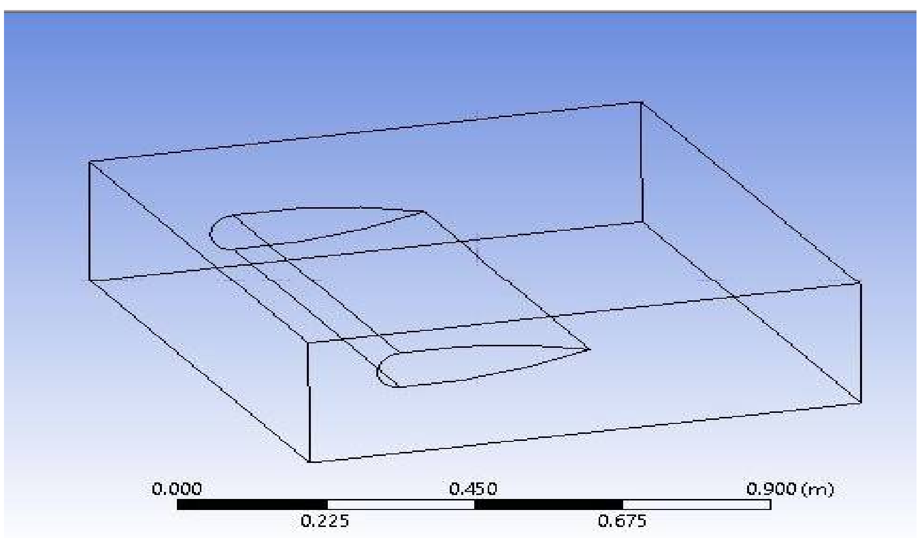

Figure 3: Spoiler geometry is imported into ansys and boundary enclosure is done (All dimensions are in m)

Step 2

After successfully finishing the geometry, it is imported to ICEM CFD for meshing. In order to obtain fine mesh unstructured mesh is done with tetrahedral cell shape as shown in figure 4. Tetrahedron has 6 edges, 4 vortices and also bounded with 4 triangular faces. The mesh contains 18567 elements and 55793 nodes.

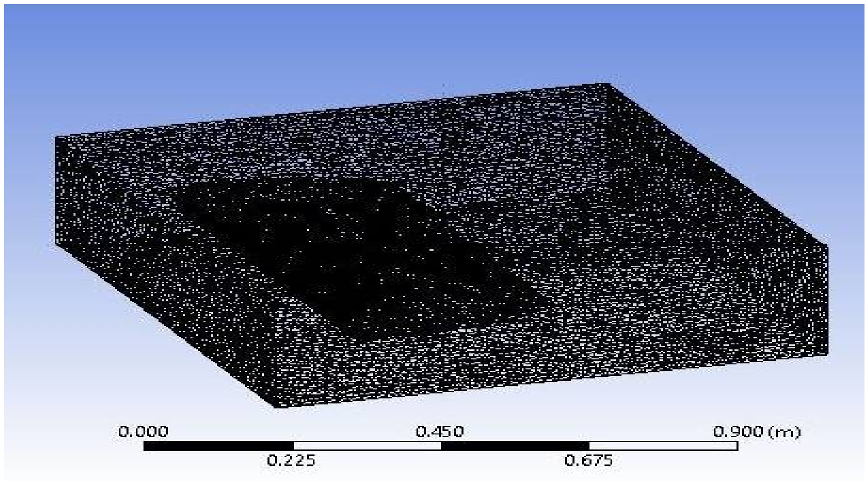

Figure 4: Unstructured Mesh of Spoiler - Tetrahedron Cell shape (All dimensions are in m)

5. Results and Discussion

Analaytical result of airfoil shaped spoiler facing inlet velocity of 30 m/s is shown below, the pressure distribution over 2D airfoil is shown in figure 6. and it is predicted that lower pressure in upper surface and high pressure in lower surface of the airfoil shaped spoiler and the pressure is maximum at its stagnation point. Figure 7. Shows the velocity contour of 2D airfoil and the analyzing results shows high velocity in upper surface of the airfoil due to low pressure distribution and vice versa in lower surface of the airfoil.

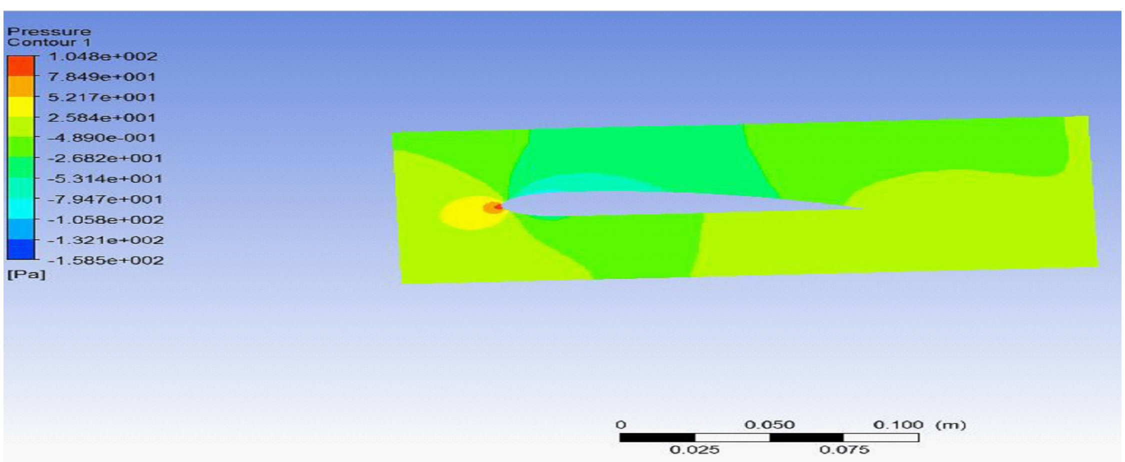

Figure 6: Pressure distribution over the cross section of spoiler (All dimensions are in m)

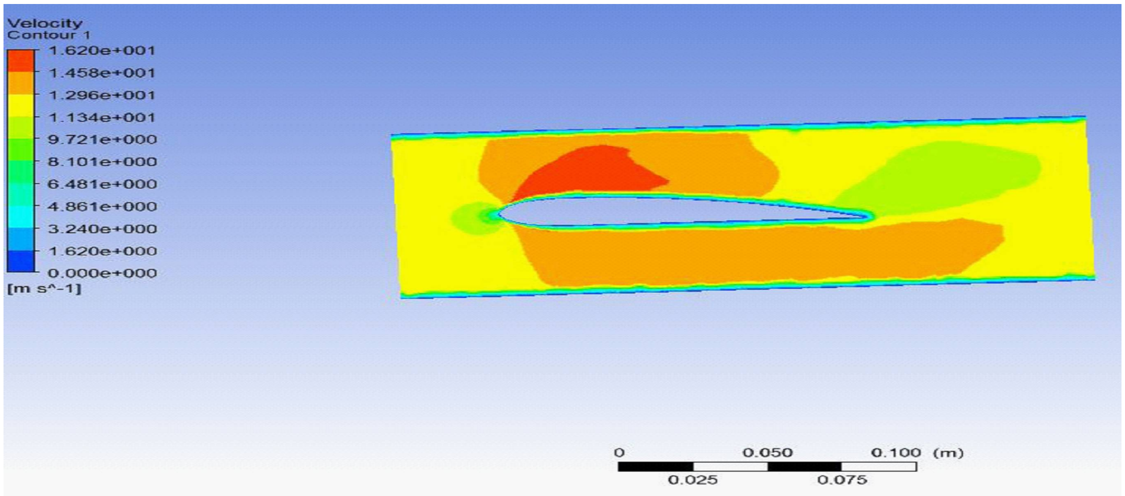

Figure 7: Velocity distribution over the cross section of spoiler (All dimensions are in m)

Figure 8 shows the pressure distribution of 3 dimensional airfoil shaped spoiler and the results shows the spoiler device can produce enough lift and reduces the drag due to the lower pressure over the surface and high pressure under the surface. The aerodynamic force like lift and drag will be acting at centre of pressure. The computational results shows that airfoil shaped spoiler are having lower drag coefficient which will be useful for fuel economy comparing to ordinary commercial and other shaped spoilers. Then a prototype scaled model of airfoil shaped spoiler is fabricated using wood for wind tunnel testing to verify the computational result.

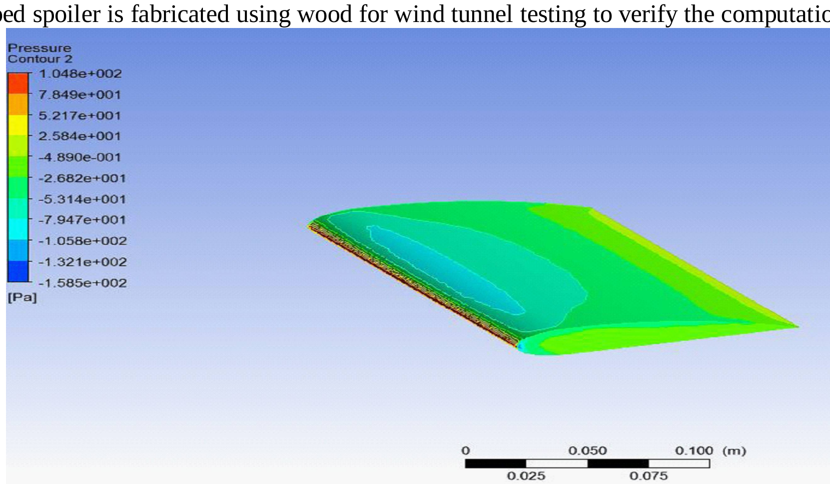

Figure 8: Pressure distribution over the full span of airfoil shaped spoiler (All dimensions are in m)

Wind tunnel testing was applied to aeronautical, mechanical, and automobiles not so much but to determine aerodynamic forces and to determine ways how to reduce the power required to speedup the vehicle on roadways with higher speed. In these cases, the interaction between the vehicle and the road plays very significant role, and this interaction should be taken into the consideration when interpreting the obtained test results. An actual situation the road are moving relative to the vehicle but the air is always stationary relative to the roadway, but when it comes to the wind tunnel testing the air is moving relative to the roadway, while the roadway is kept stationary relative to test the vehicle. Some automobile vehicle-test wind tunnels have already incorporated moving belts under the test vehicle in an effort to make an approximate of the actual condition, and very similar device were used in wind tunnel testingrelated to aircraft take-off and landing configurations.

The scaled prototype model of airfoil shaped spoiler is kept inside a wind tunnel test section of dimensions 100 cm X 45 cm and then the experimental test begins and the readings are noted down for various the angle of attack 0^0, 5^0, 10^0,15^0 for input velocity of 30 m/s. Table 1 shows the experimental analysis of airfoil shaped spoiler.

The advancement in computational fluid dynamics (CFD) modeling on high speed digital computer had reduced the demand for wind tunnel vehicle testing. However, CFD results were still not completely reliable and it is necessary for wind tunnel test are used to verify CFD predictions.

Figure 9: Wind Tunnel testing of Airfoil shaped spoiler scaled model

Table 1: Experimental results of Airfoil Shaped Spoiler

Sl. No.	Frequency	Angle of Attack	Velocity (m/s)	Co - Efficient of lift C_L	Co – Efficient of Drag C_D
1	25	0	30	0.062	0.0314
2	25	5	30	0.5443	0.2721
3	25	10	30	0.1087	0.054
4	25	15	30	0.024	0.024

6. Conclusions

Computational fluid dynamics (CFD) solutions of the steady flow field condition around passenger car models with fixed airfoil shaped Spoiler were presented and compared with the experimental data. The ANSYS-16.0 Fluent with the k-ε (epsilon) steady model is used for acquiring the simulations of aerodynamics while using rear spoiler in automobile vehicles. In this analysis, the coefficient of drag has been siginificantly reduced and coefficient of lift is also reduced. Hence, the airfoil shaped Spoiler is the effective tool to reduce the drag force on automobile car. The effects of different aerodynamic spoiler devices over flow conditions and its structure over a passenger car can be analyzed using CFD approach. The main objective is to reduce aerodynamic drag acting on the automobile vehicle can be achieved and thus to improve the fuel efficiency of passenger car. Hence, the opposite air resistance drag force can be reduced by using airfoil shaped spoiler devices on vehicle and fuel economy and stability of a passenger car can be improved.

7. References

[1] Nilanjan Saha, Gap Size Effect on Low Reynolds Number Wind Tunnel Experiments.

[2] Jeppe Johansen and Niels N. Sørensen, Numerical Analysis of winglets on Wind Turbine Blades Using CFD.

[3] K. Kondo, the Wall Interference of Wind Tunnels with Boundaries of Circular Arcs, Tokyo Imperial University, Aeronautical Research Institute, Report No. 126, 1935.

[4] K. Kondo, Boundary Interference of Partially Closed Wind Tunnels, Tokyo Imperial University, Aeronautical Research Institute, Report No. 137, 1936.

[5] E. Pistolesi on the Interference of a Wind Tunnel with a Mixed Boundary, Pont. Acad. Sci.4, No. 9 (1940). Translated by Cornell Aeronautical Laboratory, Inc., 1949).

[6] E. Pistolesi, the Interference Effect of a Wind Tunnel with Mixed Contour, Pont. Acad. Sci.7, No. 5 (1943). (Translated by Langley Aero. Lab., NACA 1951)

[7] R. H. Wright and V. G. Ward ,NACA Transonic Wind Tunnel Test Section, NACA RM L8J06, October 1948.

[8] D. D. Davis, j r., and D. Moore, Analytical Study of Blockage- and Lift-Interference Corrections for Slotted Tunnels Obtained by the Substitution of an Equivalent Homogeneous Boundary for the Discrete Slots, NACA RM L53 E07b, June 1953

Author Biography

PON MAA KISHAN A is pursuing PhD degree at Regional Campus Anna University Tirunelveli, India. He received his Master degree in Aeronautical Engineering from Anna University Chennai, India in the year 2015. His area of interest is Aerodynamics, Aircraft Structure and Space Mechanics. He is also a Member in International Association for Engineers, Hongkong.