American Journal of Engineering Research (AJER)
e-ISSN: 2320-0847 p-ISSN : 2320-0936
Volume-6, Issue-8, pp-338-344
www.ajer.org
Research Paper Open Access

Numerical Investigation of the Fluid Flow Characteristics on the Surface of an Airfoil with Different Angles of Vortex Generator's

Mohammad IliasInam*, Mohammad Mashud
Department of Mechanical Engineering,Khulna University of Engineering & Technology, Khulna, Bangladesh
Corresponding Author: Mohammad IliasInam

Abstract : *The goal of this endeavor is to analyze the variation of flow physics of flow over an airfoil without spoiler and with spoiler at different angles. From this analysis how stall point are decreased with spoiler angle variation are to be found. The coefficient of pressure, counter display, vector display are observed for all the different mesh design. For different values of spoiler angle, the variation of lift and drag component of force can be analyzed. The general-purpose CFD software FLUENT is used for this observation. Fluent employs experts in computational methods, mesh generation, and software development. All functions required to compute a solution and display the results are accessible in FLUENT through an interactive, menu-driven interface. That's why it is used for this analysis. At first the design of airfoil without spoiler and with spoiler at different angle is drawn in GAMBIT 2D simulation. The problem specifications are as follows- create geometry in GAMBIT, mesh geometry in GAMBIT, specify boundary types in GAMBIT, set up problem in FLUENT, solve the problem and analyze the results. This investigation led to a conclusion that lift is decreased and drag is increased which shows the common nature of spoiler.*

Keywords: *Fluid Flow, Aerodynamics, Airfoil and Vortex Generator.*

Date of Submission: 17-08-2017 Date of acceptance: 31-08-2017

I. INTRODUCTION

The cross-sectional shape obtained by the intersection of the wing of an airplane with the perpendicular plane is called an airfoil [1]. There is an aerodynamic force created by the pressure and shear stress distributions over the wing surface. This resultant aerodynamic force can be resolved into two forces, parallel and perpendicular to the relative wind. The direction of free stream velocity is defined as the relative wind. The two forces are called lift and drag force. The drag is always defined as the component of the aerodynamic force parallel to the relative wind. The lift is defined as the component of the aerodynamic force perpendicular to the relative wind.As a wing moves through air, the air is split and passes above and below the wing. The wing's upper surface is shaped so the air rushing over the top speeds up and stretches out. This decreases the air pressure above the wing. The air flowing below the wing moves in a straighter line, so its speed and air pressure remain the same. Since high air pressure always moves toward low air pressure, the air below the wing pushes upward toward the air above the wing. The wing is in the middle, and the whole wing is "lifted." The faster an airplane moves, the more lift there is. And when the force of lift is greater than the force of gravity, the airplane is able to fly.In this project work, the commercial tool FLUENT is used. For mesh geometry the GAMBIT 2D simulation is used. The commercial package FLUENT is a powerful and flexible general-purpose CFD software developed by ANSYS, Inc. Thousands of companies throughout the world benefit from this engineering design and analysis tool, using FLUENT for a wide variety of multiphysics applications. At first the design of airfoil without spoiler and with spoiler at different angle is drawn in GAMBIT 2D simulation. The problem specifications are as follows- create geometry in GAMBIT, mesh geometry in GAMBIT, specify boundary types in GAMBIT, set up problem in FLUENT, solve the problem and analyze the results.For this research NACA 2415 airfoil was used.

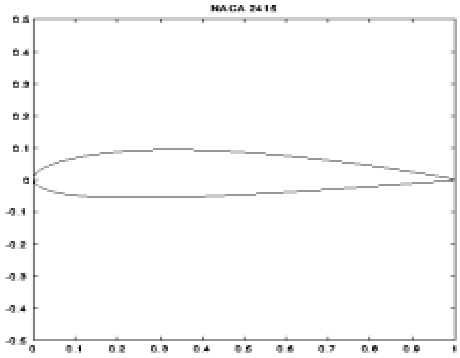


Figure 1: profile of NACA 2415 airfoil

A spoiler, sometimes called a lift dumper is a device intended to reduce lift in an aircraft. Spoilers are plates on the top surface of a wing which can be extended upward into the airflow and spoil it. By doing so, the spoiler creates a carefully controlled stall over the portion of the wing behind it, greatly reducing the lift of that wing section. Spoilers are designed to reduce lift also making considerable increase in drag [2].Spoilers are used by some older gliders to control their rate of descent and thus achieve a controlled landing at a desired spot. An increased rate of descent could also be achieved by lowering the nose of an aircraft, but this would result in an excessive landing speed. However spoilers enable the approach to be made at a safe speed for landing.Airliners too are usually fitted with spoilers. Spoilers are sometimes used when descending from cruise altitudes to assist the aircraft in descending to lower altitudes without picking up speed. Their use is often limited, however, as turbulent airflow which develops behind them causes noticeable noise and vibration, which may cause discomfort to extra-sensitive passengers. The spoilers may also be differentially operated to provide roll control. Martin Aircraft was the first to develop spoilers to help with roll control in 1948. On landing , however, the spoilers are nearly always used at full effect to assist in slowing the aircraft. The increase in form drag created by the spoilers directly assists the braking effect. However, the real gain comes as the spoilers cause a considerable loss of lift and hence the weight of the aircraft is transferred from the wings to the undercarriage, allowing the wheels to be mechanically braked with much less chance of skidding [2].Spoilers increase drag and reduce lift on the wing. If raised on only one wing, they aid roll control, causing that wing to drop. If the spoilers raise symmetrically in flight, the aircraft can either be slowed in level flight or can descend rapidly without an increase in airspeed. When the spoilers rise on the ground at high speeds, they reduce the wing's lift, which puts more of the aircraft's weight on the wheels. The flight spoilers are available both in flight and on the ground. However, the ground spoilers can only be raised when the weight of the aircraft is on the landing gear, usually activated by a sensor. When the spoilers deploy on the ground, they decrease lift and make the brakes more effective. In flight, a ground-sensing switch on the landing gear prevents deployment of the ground spoilers.The objectives of this research works are: to Study the FLUENT and GAMBIT Commercial simulation tool and observe the effect of spoiler on aerodynamic characteristics of an airfoil with the variation of spoiler angle at leading edge.

II. METHEMATICAL MODELING

Two equation turbulence models are one of the most common type of turbulence models. Models like the k-epsilon model and the k-omega model have become industry standard models and are commonly used for most types of engineering problems. Two equation turbulence models are also very much still an active area of research and new refined two-equation models are still being developed. By definition, two equation models include two extra transport equations to represent the turbulent properties of the flow. This allows a two equation model to account for history effects like convection and diffusion of turbulent energy [3]. Most often one of the transported variables is the turbulent kinetic energy K.The second transported variable varies depending on what type of two-equation model it is. Common choices are the turbulent dissipation ε, or the specific dissipation ω. The second variable can be thought of as the variable that determines the scale of the turbulence (length-scale or time-scale), whereas the first variableK, determines the energy in the turbulence.The K-epsilon model is one of the most common turbulence models, although it just doesn't perform well in cases of large adverse pressure gradients. It is a two equation model that means, it includes two extra transport equations to represent the turbulent properties of the flow. This allows a two equation model to account for history effects like convection and diffusion of turbulent energy. The first transported variable is turbulent kinetic energy, K. The second transported variable in this case is the turbulent dissipation, ε. It is the variable that determines the scale of the turbulence, whereas the first variable K, determines the energy in the turbulence.There are two major formulations of K-epsilon models . The original impetus for the K-epsilon model was to improve the mixing-length model, as well as to find an alternative to algebraically prescribing turbulent length scales in moderate to high complexity flows. The K-epsilon model has been shown to be useful for free-shear layer flows with relatively small pressure gradients. Similarly, for wall-bounded and internal flows, the model gives good results only in cases where mean pressure gradients are small; accuracy has been shown experimentally to be reduced for flows containing large adverse pressure gradients. One might infer then, that the K-epsilon model would be an inappropriate choice for problems such as inlets and compressors [3].The k-ε model introduces two new variables into the system of equations. The **continuity equation** is then:

$$\frac{\partial \rho}{\partial t}+\frac{\partial}{\partial x_j}\left(\rho U_j\right)=0$$

And the **momentum equation** becomes:

$$\frac{\partial \rho U_i}{\partial t}+\frac{\partial}{\partial x_j}\left(\rho U_i U_j\right)=-\frac{\partial p'}{\partial x_i}+\frac{\partial}{\partial x_j}\left[\mu_{eff}\left(\frac{\partial U_i}{\partial x_j}+\frac{\partial U_j}{\partial x_i}\right)\right]+S_M$$

A. *Transport equations:*

$$\frac{\partial}{\partial t}(\rho k)+\frac{\partial}{\partial x_j}(\rho k u_j)=\frac{\partial}{\partial x_j}\left[\left(\mu+\frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right]+P_k+P_b-\rho\epsilon-Y_M+S_k$$

$$\frac{\partial}{\partial t}(\rho\epsilon)+\frac{\partial}{\partial x_j}(\rho\epsilon u_j)=\frac{\partial}{\partial x_j}\left[\left(\mu+\frac{\mu_t}{\sigma_\epsilon}\right)\frac{\partial \epsilon}{\partial x_j}\right]+\rho C_1 S\epsilon-\rho C_2\frac{\epsilon^2}{k+\sqrt{\nu\epsilon}}+C_{1\epsilon}\frac{\epsilon}{k}C_{3\epsilon}P_b+S_\epsilon$$

Where

$$C_1=\max\left[0.43,\frac{\eta}{\eta+5}\right],\qquad \eta=S\frac{k}{\epsilon},\qquad S=\sqrt{2S_{ij}S_{ij}}$$

In these equations, $P_k$represents the generation of turbulence kinetic energy due to the mean velocity gradients, calculated in same manner as standard k-epsilon model. $P_b$is the generation of turbulence kinetic energy due to buoyancy, calculated in same way as standard k-epsilon model.

B. *Modeling Turbulent Viscosity*

$$\mu_t=\rho C_\mu\frac{k^2}{\epsilon}$$

Where

$$C_\mu=\frac{1}{A_0+A_s\frac{kU^*}{\epsilon}}$$

$$U^*\equiv\sqrt{S_{ij}S_{ij}+\tilde{\Omega}_{ij}\tilde{\Omega}_{ij}};$$

$$\tilde{\Omega}_{ij}=\Omega_{ij}-2\epsilon_{ijk}\omega_k;$$

$$\Omega_{ij}=\overline{\Omega_{ij}}-\epsilon_{ijk}\omega_k$$

where $\overline{\Omega_{ij}}$ is the mean rate-of-rotation tensor viewed in a rotating reference frame with the angular velocity ω_k. The model constants A_0 and $A_s$are given by:

$$A_0=4.04,\quad A_s=\sqrt{6}\cos\phi$$

$$\phi=\frac{1}{3}\cos^{-1}(\sqrt{6}W),\quad W=\frac{S_{ij}S_{jk}S_{ki}}{\tilde{S}^3},\quad \tilde{S}=\sqrt{S_{ij}S_{ij}},\quad S_{ij}=\frac{1}{2}\left(\frac{\partial u_j}{\partial x_i}+\frac{\partial u_i}{\partial x_j}\right)$$

C. *Model Constants*

$$C_{1\epsilon}=1.44,\quad C_2=1.9,\quad \sigma_k=1.0,\quad \sigma_\epsilon=1.2$$ [2]

III. SOLUTION METHOD

The solution method utilized for the simulation has a pressure based solver with implicit formulation, 2-D domain geometry, absolute velocity formulation, and superficial velocity for porous formulation. For this test, a simple solver and an external compressible flow model for the turbulence was utilized. The green-gauss cell based was used for the gradient option. There are different equations used for flow and turbulence. A simple method was used for the pressure-velocity coupling. For the discretization, a standard pressure was used, and Density, momentum and turbulent kinetic energy were set to first order upwind.

Inlet velocity for the experiments and simulations is 5m/sec and turbulence viscosity ratio is 10. A fully turbulent flow solution was used in ANSYS fluent 6.3.26, where k-ε model was used for turbulent viscosity. A simple solver was utilized .The airfoil profile and boundary conditions are all created by using another software named Gambit 2.4.6.The mesh generation was done by this software.

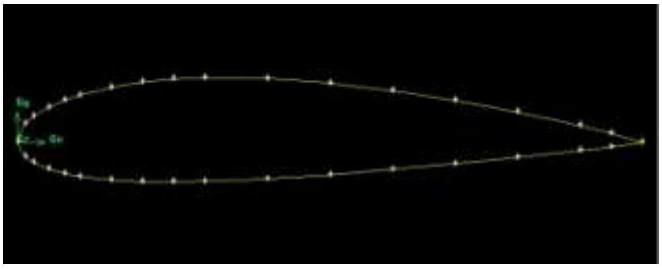
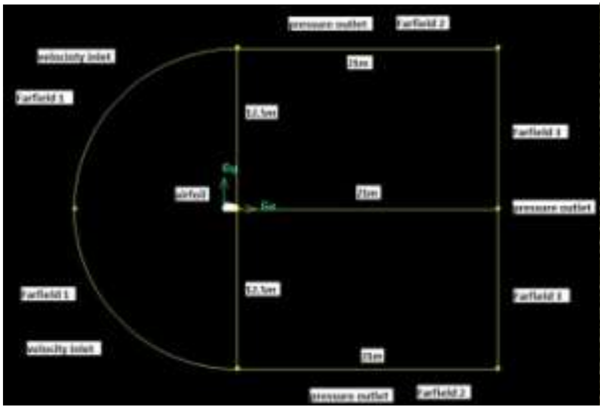


Figure 2: Airfoil geometry created in Gambit NACA 2415



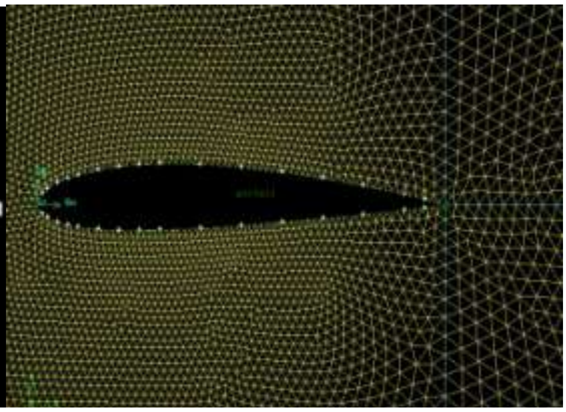


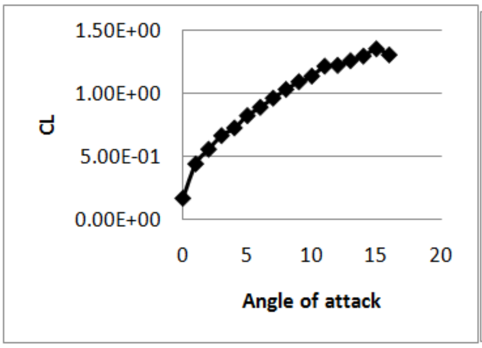
Figure 3: Computational domain and boundary conditions.**Figure4:** Mesh generated for simulation

IV. RESULTS AND DISCUSSION

Lift and drag force component have been considered as the governing physical parameter in this problem. All the values related to lift and drag force have been varied over wide ranges to study the effects on the graph of lift coefficient or drag coefficient versus angle of attack, coefficient of pressure. To study figure of velocity counter, grid. In the present investigation, a mesh file has been created by GAMBIT software and read by FLUENT. The range of angle of attack is 0 to 15^0. For little range of iteration the constant value of coefficient of lift (C_L)can be obtained easily. After completing the iteration for each value of angle of attack(α), a constant value of C_L has been obtained. For this range of angle of attack, the graph of coefficient of lift (C_L) versus angle of attack (α) has been developed. From where a stall point has been obtained for a fixed value of C_L. The same procedures have been repeated for different mesh files of different spoiler angle. Using a single mesh file, velocity convergent graph, C_L Vs α graph, pressure coefficient (C_P) graph, figure of vector,counter,grid can be obtained.

In case of drag component, same conditions has been used. After completing the iteration for each value of angle of attack(α), a constant value of coefficient of drag (C_D)has been obtained. For this range of angle of attack, the graph of coefficient of drag (C_D) versus angle of attack (α) has been developed. The graph obtained for drag is directing upward and very smooth.After completing iteration, for each value of angle of attack from the graph of C_LVs iteration a contant value of C_L has been obtained. By plotting all the values of C_L for all values of α a graph can be obtained.

For without spoiler, a stall point has been got at α=15 which is given below:



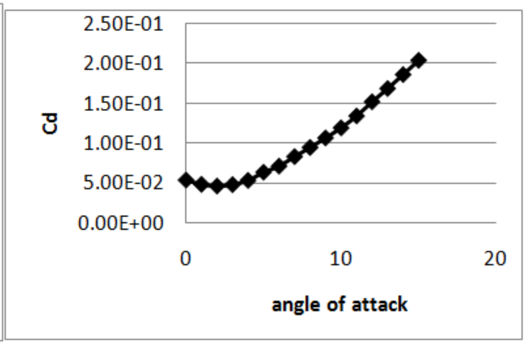
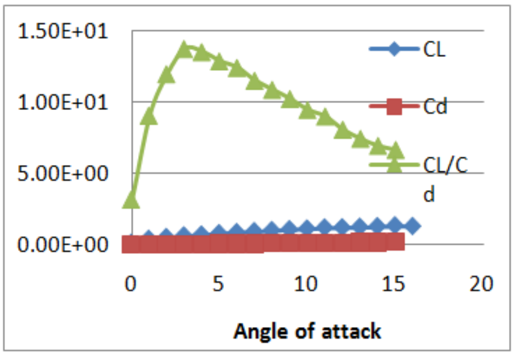


Figure 5: C_L Vs α for without spoiler**Figure 6:** C_D Vs α for without spoiler

By the procedure described above, in case of drag component after plotting all the values of C_D for all values of α a graph can be obtained which is given below:By plotting C_L/C_DVs α for without spoiler also has been got,



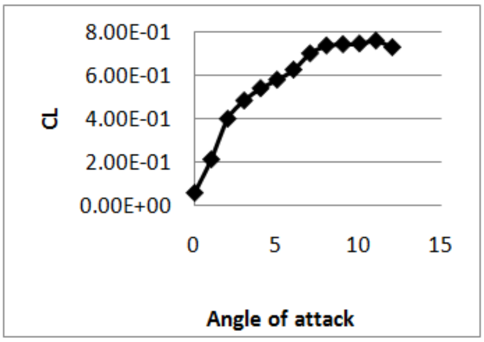


Figure 7: C_L/C_DVs α for without spoiler**Figure 8:** C_L Vs α with spoiler angle = 2.5

A airfoil with spoiler angle = 2.5 has been designed for NACA 2415. The figure of grid is given below which is common for this design:After completing iteration, for each value of angle of attack from the graph of C_LVs iteration a contant value of C_L has been obtained. By plotting all the values of C_L for all values of α a graph can be obtained. For spoiler angle = 2.5, a stall point has been got at α =11 which is shown in figure 7 and 8For spoiler angle 2.5, 5, 7.5, 10 and 12.5, the graphsare show in the next section

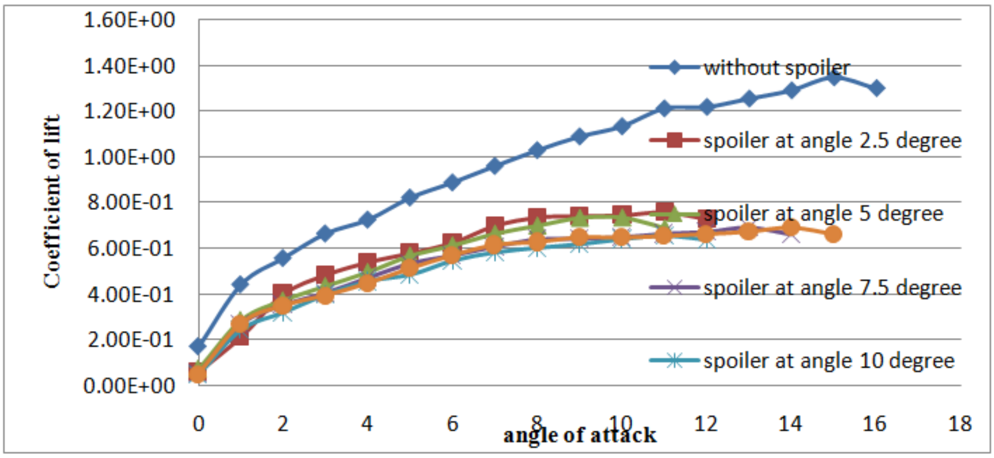


Figure 143: Comparison of the C_l vs. angle of attack curve for airfoil with spoiler to the airfoil without spoiler

From the figure above we have for the airfoil having spoiler placed in any particular angle has a lift curve trending lower than the airfoil without any spoiler. Thus the effect of the spoiler for all the angles are demonstrated through the figure. A certain decrement of lift coefficient is obtained and it is quite reasonable. A spoiler is used to decrease the lift of the aircraft so as to descent the aircraft. For safe and smooth landing of the airplane spoiler plays an important role by decreasing the lift coefficient thus the lift is decreased with increasing angle of spoiler. From above graph it is understandable that by attaching a spoiler to the wing section or airfoil lift obtained is lower than that of the wing section without the spoiler attachment which is one of the main functions of spoiler . So the numerical result obtained is reliable [2].

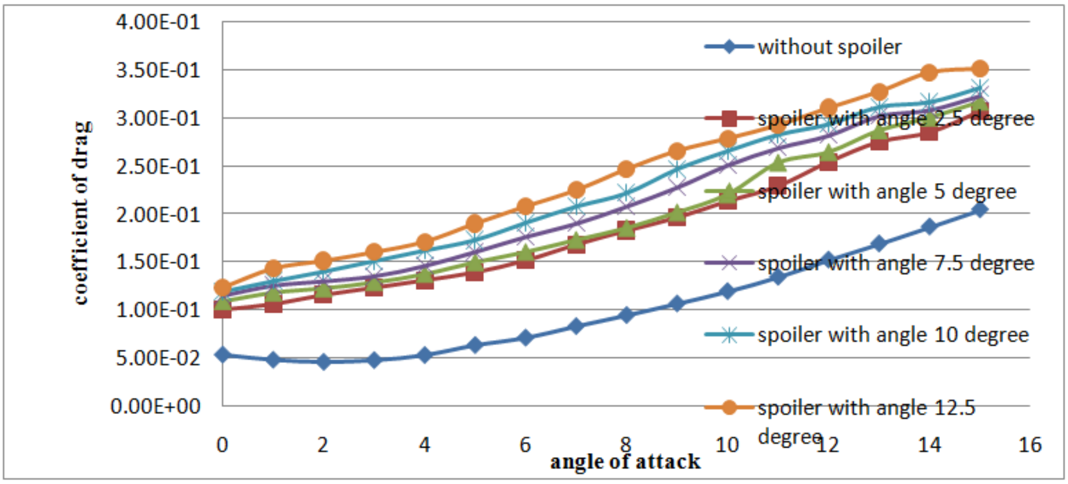


Figure 145: Comparison of the C_d vs. angle of attack curve for airfoil with spoiler to the airfoil without spoiler

From the figure above we have for the airfoil having spoiler with different angles have a drag curve trending higher than that of the airfoil without any spoiler. Thus the effect of the spoiler for all angles are demonstrated through the figure. Increased drag coefficient is the indication of increase in drag force, thus the speed of the aircraft is decreased. Before landing on the desired destination speed of the aircraft must be slowed down for safe landing. A spoiler acts as an arrangement which decreases lift by the cost of increase in drag force. From the above graph it is shown that the wing section or airfoil having spoiler attached at different angles increases drag coefficient thus the drag force than that of the airfoil having no spoiler. It is one of the principle characteristics of spoiler [12]. So the results obtained from numerical simulation are supporting the basic concepts of the spoiler.

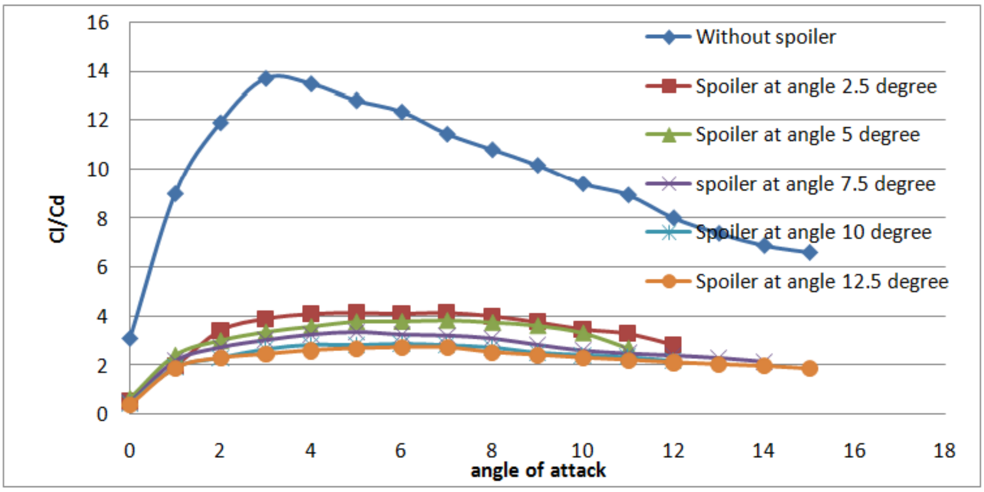


Figure 146: Comparison of the C_l/C_d vs angle of attack curve for airfoil with spoilerto the airfoil without spoiler

From the figure above we have for the airfoil having spoiler at different angles has a C_l/C_d curve trending lower than the airfoil without any spoiler. Thus the effect of the spoiler for all angles are demonstrated through the figure.

V. CONCLUSION

The mentioned problem is solved by a commercial software FLUENT. Spoiler is placed at 5 percent of chord and spoiler length is 10 percent of chord. Spoiler angles 2.5, 5, 7.5, 10, 12.5 degree are held in design.This investigation led to a conclusion that lift is decreased and drag is increased which shows the common nature of spoiler. A spoiler, sometimes called a lift dumper is a device intended to reduce lift in an aircraft. Spoilers increase drag and reduce lift on the wing. From the results discussed above it can be concluded that the numerical experiment of the NACA 2415 airfoil having spoiler on five different angles have been identical to the main function of the spoiler.

REFERENCES

[1.] M. Mashud and R. C. Das, "Effect of rear end spoiler angle of a sedan car", American Institute of Physics (AIP) Proc., ISBN: 978-0-7354-1525-6, Vol. 1851, 0200171-7; DOI: 10.1063/1.4984646, 2017.

[2] A. Y. Payl and M. Mashud, "Experimental investigation on fuel properties of biodiesel prepared from cottonseed oil", American Institute of Physics (AIP) Proc., ISBN: 978-0-7354-1525-6, Vol. 1851, 0200181-6; DOI: 10.1063/1.4984647, 2017.

[3] A. B. Shahid and M. Mashud, "Experimental and numerical study of control of flow separation of a symmetric airfoil with trapped vortex cavity", American Institute of Physics (AIP) Proc., ISBN: 978-0-7354-1525-6, Vol. 1851, pp. 0200191-6, DOI: 10.1063/1.4984648, 2017.

[4] M. R. Amin, S. M. M. Rahman, M. Mashud, and M. F. Rabbi, "Design and construction of an airfoil with controlled flap", American Institute of Physics (AIP) Proc., ISBN: 978-0-7354-1525-6, Vol. 1851, 0200901-6; DOI: 10.1063/1.4984719, 2017.

[5] M. Mashud and M. A. Hossain,"An Experimental Investigation of Active Control of Fully Seperated Flow over a Symmetrical Circular Arc Airfoil", Mechanical Engineering Research Journal (MERJ), (ISSN: 1990-5491), Vol.10, pp. 20-25, 2016.

[6] M. F. Rabbi, R. Nandi and M. Mashud,"Induce Drag Reduction of an Air plane Wing", American Journal of Engineering Research (AJER), (e-ISSN: 2320-0847 p-ISSN : 2320-0936), Vol.4, Issue-6, pp-219-223, 2015.

[7] M. F. Rabbi, M. Sarkar and M. Mashud," Alternative Room Cooling System" ,American Journal of Engineering Research (AJER), (e-ISSN: 2320-0847 p-ISSN : 2320-0936), Vol.4, Issue-6, pp-215-218, 2015.

[8] M. R. Noor, M. Assad-Uz-Zaman and M. Mashud, "Effect of Co-Flow Jet over an Airfoil: Numerical Approach", Journal of Contemporary Engineering Sciences, HIKARI Ltd., ISSN 1313-6569 (print), ISSN 1314-7641 (online), Vol. 7, no. 17, pp. 845-851, 2014.

[9] M. M. Rahman, M. T. I. Gias, M. Mashud and M. M. Islam," Mach and Pressure Distribution at the Exit of the Nozzle: A Numerical Documentation" International Journal of Engineering & Technology IJET-IJENS, Vol:14 No:03, pp. 18-27, 2014.

Mohammad IliasInam. "Numerical Investigation of the Fluid Flow Characteristics on the Surface of an Airfoil with Different Angles of Vortex Generator's." International Journal of Engineering Research and Applications (IJERA), vol. 6, no. 8, 2017, pp. 338–344.