Effect Of Spoiler Position On Aerodynamic Characteristics Of An Airfoil

Mohammad Mashud , Mausumi Ferdous and Shahriar Hossain Omee
Department of Mechanical Engineering
Khulna University of Engineering & Technology
KUET, Khulna-9203, Bangladesh
E-mail: mdmashud@yahoo.com

***Abstract*-- In this paper the effect of spoilers on aerodynamic characteristics of an airfoil were observed by CFD.As the experimental airfoil NACA 2415 was choosen and spoiler was extended from five different positions based on the chord length C. Airfoil section is designed with a spoiler extended at an angle of 7 degree with the horizontal.The spoiler extends to 0.15C.The geometry of 2-D airfoil without spoiler and with spoiler was designed in GAMBIT.The numerical simulation was performed by ANSYS Fluent to observe the effect of spoiler position on the aerodynamic characteristics of this particular airfoil. The results obtained from the computational process were plotted on graph and the conceptual assumptions were verified as the lift is reduced and the drag is increased that obeys the basic function of a spoiler.**

***Index Term*— Spoiler, Airfoil, Coefficient of lift and Coefficient of drag.**

I. INTRODUCTION

An airplane wing has a special shape called an airfoil. 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 remains 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. [1]

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. 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 rise 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. 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.

There are several mechanisms applied to the aircraft to obtain safe landing. For landing safely the speed of the aircraft must be slowed down thus the drag must be increased. Airbrakes are one of the mechanisms used to reduce the speed of the aircraft where the spoilers differ from airbrakes in that airbrakes are designed to increase drag making little change to lift, while spoilers greatly reduce lift making only a moderate change in drag. There are several other benefits of occupying spoiler on a aircraft wing. Rapid descents may be made without having to reduce power, thereby maintaining engine temperatures at a comfortable level, and eliminating the risk of engine "shock cooling." Spoilers can maintain normal cruise/descent speeds until much closer to the airport without worrying about how to slow to gear extension speed and deploying spoilers instead of reducing power and/or lowering the landing gear will safely accomplish rapid reduction in airspeed. The aircraft can maintain a high altitude flight longer and take the advantage of tailwinds and smooth air until the aircraft is much closer to the destination before beginning the descent.

This paper mainly focuses on the effect of spoiler on the aerodynamic forces of the airfoil with a view to demonstrate the effect numerically. NACA 2415 airfoil is considered for the observation of the effect of spoiler positions on the aerodynamic characteristics of this particular airfoil. Coefficient of drag, coefficient of lift was calculated thus the characteristics curves can be drawn against the angle of attack varying from 0 to 15 and compared to the characteristics of the NACA 2415 airfoil having no spoiler. Spoilers are fitted in 5 different positions on NACA 2415 airfoil for the experiment. As the chord length of the airfoil is considered to be 1m, the airfoil is designed with the spoilers at 0.5C, 0.6C, 0.7C, 0.8C.0.9C. The spoiler is extended upwards till 0.15 m at an angle of 7 degree for all the positions.

There are a huge number of numerical investigations carried out on the high lift devices or flow separation control methods. Most of the experiments held were feasible for the utilization in the development of wing design or airfoil shape. The purpose of this paper is to demonstrate the effect of spoiler on the airfoil through numerical approach. Gambit 2.4.6 was used for the designing and generation of mesh, also for applying the boundary conditions. ANSYS fluent 6.3.26 version was used for CFD simulation. Airfoil coordinates were taken from UIUC Airfoil data site. ANSYS fluent is reliable simulation software as many of the engineering work have been previously done by this particular program which was realistic. Many computational problems were designed and solved through this program as it is easy to understand and its application is preferable than other simulation software's. The work has been divided in two parts where the first part was to create the geometry of the airfoil, mesh generation and applying the boundary conditions. The second part was the computational part and was occupied by grid checking, implementation of the solver, boundary conditions and finding the values of desired force coefficients.

II. MATHEMATICAL MODELING

Two equation turbulence models are one of the most common types 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 variable K, 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.

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:

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

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

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

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.

Modeling Turbulent Viscosity

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

Where

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

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

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

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

whe __ 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:

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

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

Where the model constants are

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

III. NUMERICAL MODELING

The solution method utilized for the simulation had 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 simulations is 5m/sec and turbulence viscosity ratio is 10. A fully turbulent flow solution was used in ANSYS fluent6.3.26, where realizable k-ε model was used for turbulent viscosity. A simple solver was utilized for the simulation.

Computational Domain:

- Airfoil – NACA 2415
- Chord length (C) 1m
- Length of the spoiler(spoiler extension) 15 cm
- Width of the spoiler 2 cm
- Position of the spoiler (spoiler attachment) at 0.5C , 0.6C , 0.7C , 0.8and 0.9C
- Angle of spoiler extension with the horizontal 7 degree
- Velocity of air (relative) 5 m/s

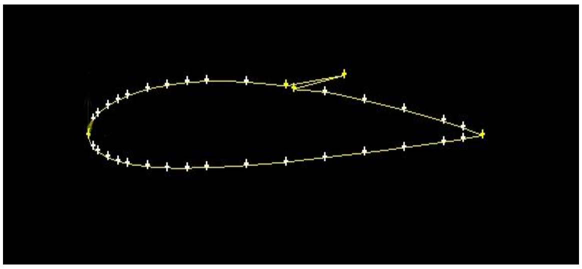

Fig. 1. Geometry of airfoil with spoiler.

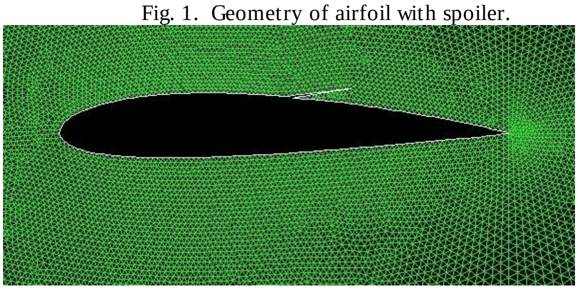

Fig. 2. Grid generated

Domain Extents:
x-coordinate: min (m) = -1.150000e+001, max (m) = 2.100000e+001
y-coordinate: min (m) = -1.250000e+001, max (m) = 1.250000e+001
Volume statistics:
minimum volume (m3): 1.347452e-005
maximum volume (m3): 3.856536e-001
total volume (m3): 6.561468e+002
Face area statistics:
minimum face area (m2): 3.271446e-003
maximum face area (m2): 9.874787e-001

The computational domain was created with 5 face zones and 86460 nodes.The domain is divided in four parts for applying the boundary conditions.These are the airfoil section, farfield 1, farfield 2, farfield 3.For the airfoil section the solid wall no slip condition was applied.For the farfield 1 boundery condition was velocity inlet and for other two farfield pressure outlet was selected as the boundery condition.The boundery condition for farfield was applied as the velocity components.For X component velocity was applied as 5cos α as α is the angle of attack.For Y component of velocity 5sin α is applied.

For the design, mesh generation and applying the boundery condition to the domain to be calculated Gambit was used and Fluent was used as the solver.Fluent has a reliable computational accuracy for fluid flow arrangements and holds good results.

For five different spoiler arrangement the design was created in Gambit and the mesh was created as per the spoiler position.Around the airfoil section the mesh was fine enough to observe the velocity and pressure contour perfectly.The effect of the spoiler on pressure distribution of airfoil surface can be observed simultaneously as the computation for each angle of attack is done.The grid generated was checked before the computation started and there was a satisfactory output for grid checking for every spoiler position.Convergence criteria was selected as 10^{-3}.This indicates the value taken as the result were constant for consecutive 1000 iterations.Other criterions like contnuity residuals were also monitored.

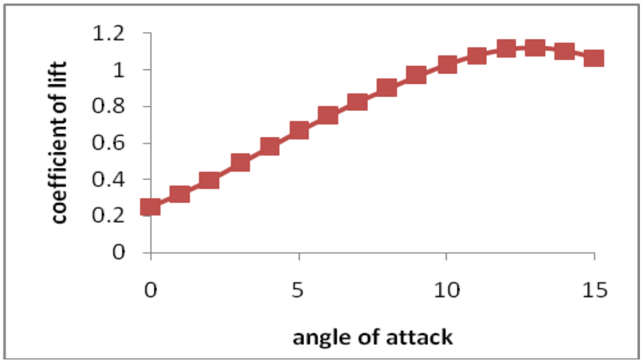

Fig. 3. Lift curve for airfoil without spoiler

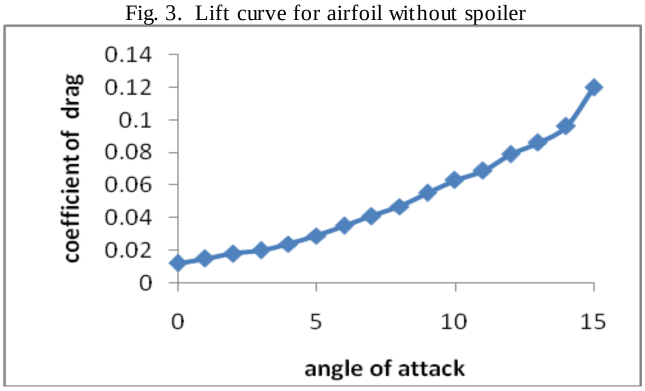

Fig. 4. Drag curve for airfoil without spoiler

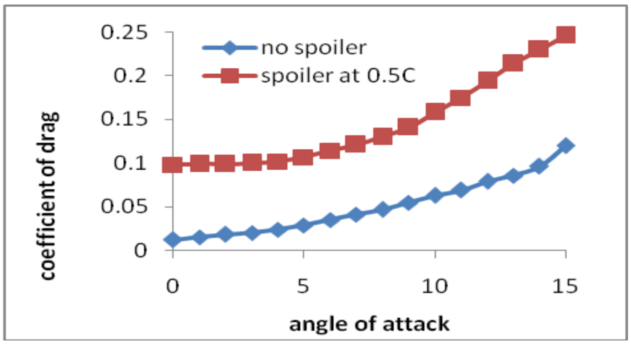

Fig. 5. Drag curve with spoiler at 0.5C compared to drag curve without spoiler.

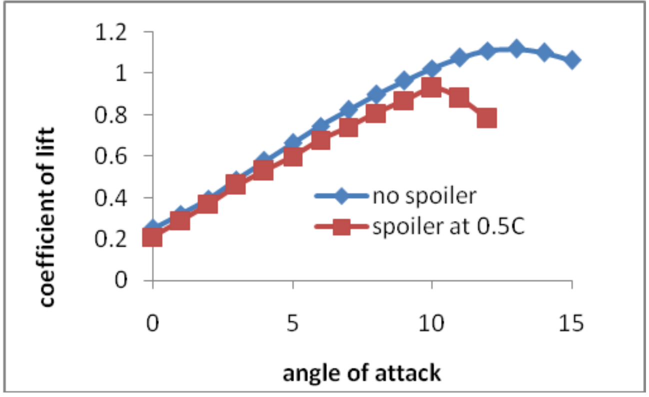

Fig. 6. Lift curve with spoiler at 0.5C compared to lift curve without spoiler.

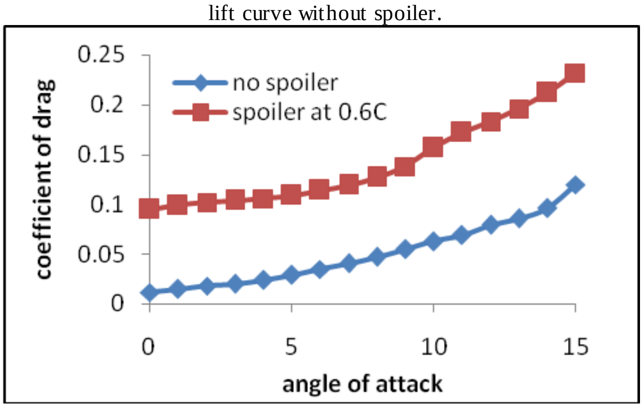

Fig. 7. Drag curve with spoiler at 0.6C compared to drag curve without spoiler.

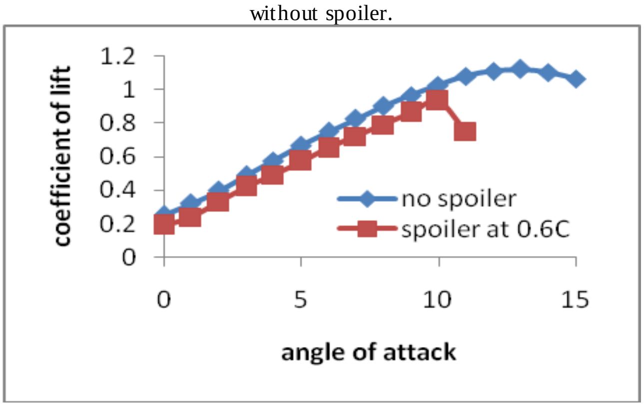

Fig. 8. Lift curve with spoiler at 0.6C compared to lift curve without spoiler

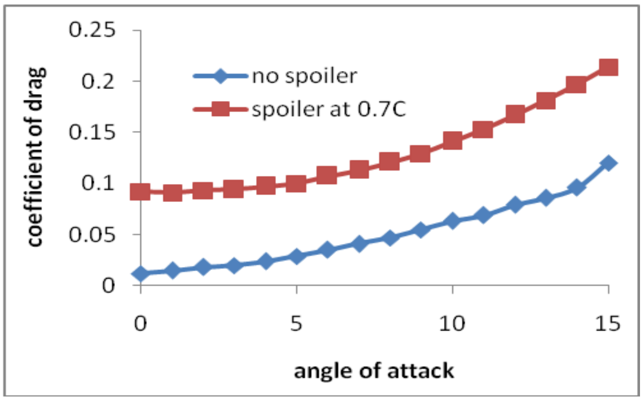

Fig. 9. Drag curve with spoiler at 0.7C compared to drag curve without spoiler

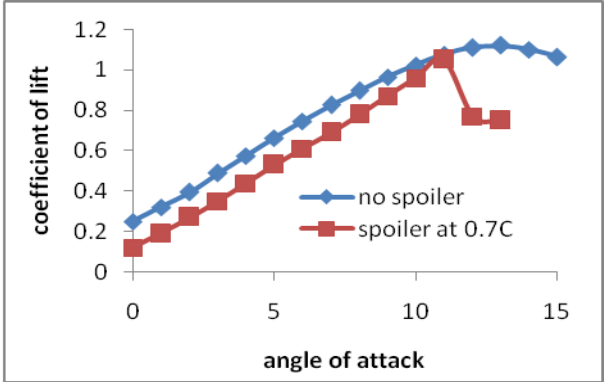

Fig. 10. Lift curve with spoiler at 0.7C compared to lift curve without spoiler

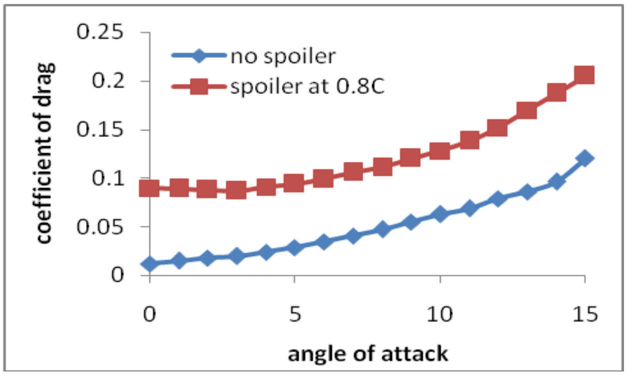

Fig. 11. Drag curve with spoiler at 0.8C compared to drag curve without spoiler

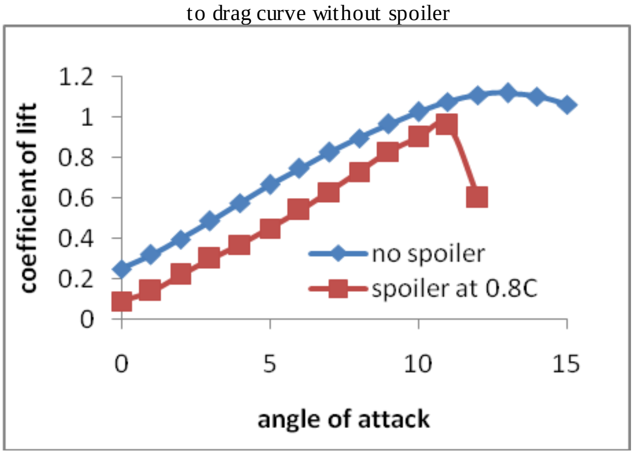

Fig. 12. Lift curve with spoiler at 0.8C compared to lift curve without spoiler

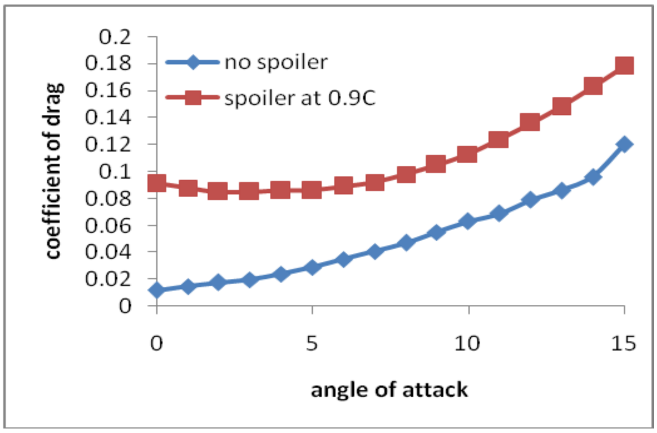

Fig. 13. Drag curve with spoiler at 0.9C compared to drag curve without spoiler

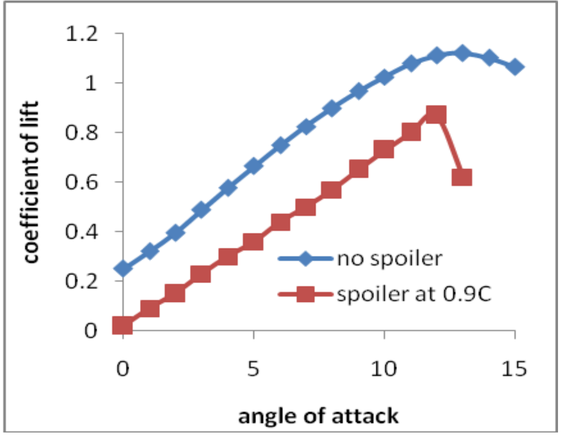

Fig. 14. Lift curve with spoiler at 0.9C compared to lift curve without spoiler

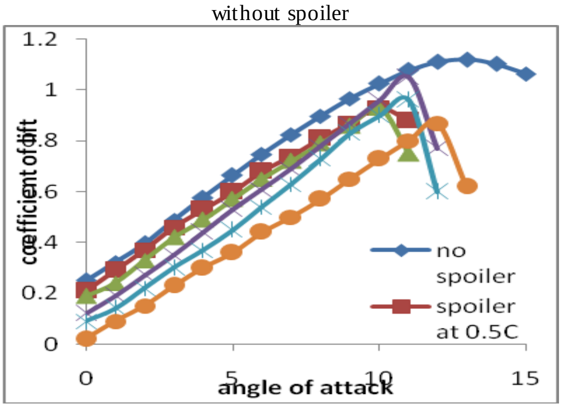

Fig. 15. Comparison of the C_l vs. angle of attack curve for airfoil with spoiler to the airfoil without spoiler

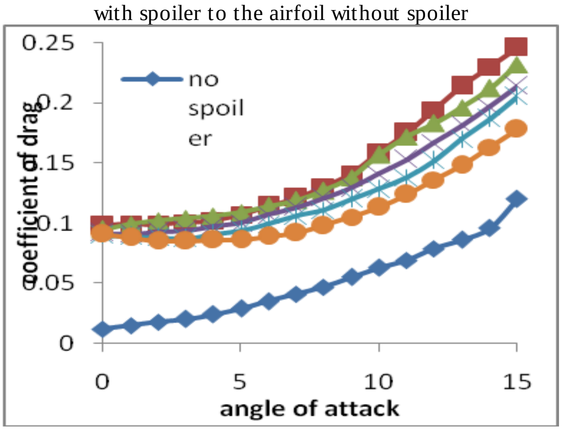

Fig. 16. Comparison of the C_d vs. angle of attack curve for airfoil with spoiler to the airfoil without spoiler

I. RESULTS AND DISCUSSION

A spoiler 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. If raised on only one wing, they aid roll control, causing that wing to drop. On some airplanes, spoilers are deployed from the wings to spoil the smooth airflow, reducing lift and increasing drag. Spoilers are used for roll control on some aircraft, one of the advantages being the elimination of adverse yaw. To turn right, for example, the spoiler on the right wing is raised, destroying some of the lift and creating more drag on the right. The right wing drops, and the airplane banks and yaws to the right. Deploying spoilers on both wings at the same time allows the aircraft to descend without gaining speed. Spoilers are also deployed to help shorten ground roll after landing. By destroying lift, they transfer weight to the wheels, improving braking effectiveness. By the figures above it is demonstrated that the airfoil having spoiler placed in any particular position has a lift curve trending lower than the airfoil without any spoiler. A certain decrement of lift coefficient is obtained and it is quite reasonable. A spoiler is used to decrease the lift and increase the drag force of the aircraft so as to descent and slow down the aircraft. For safe and smooth landing of the airplane spoiler plays an important role by decreasing the lift coefficient thus the lift force. Thus the effect of the spoiler for all the positions is demonstrated through the fig.. The numerical results obtained from the computation carried out on the effect of spoiler on aerodynamic characteristics of NACA 2415 are showing reliable agreement to the basic functions of a spoiler of an aircraft wing.

Contour of velocity magnitude and static pressure:

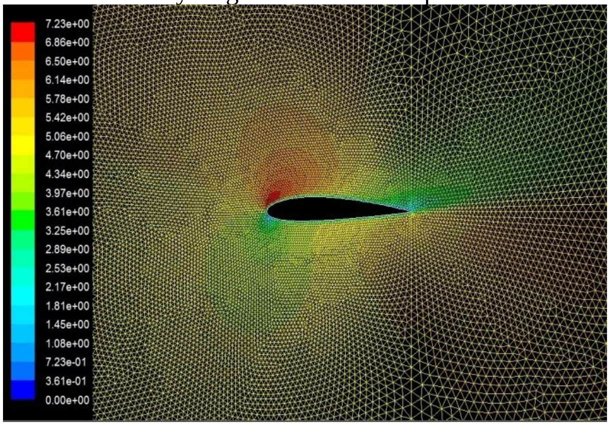

Fig. 17. Contour of velocity magnitude for $\alpha=15^0$

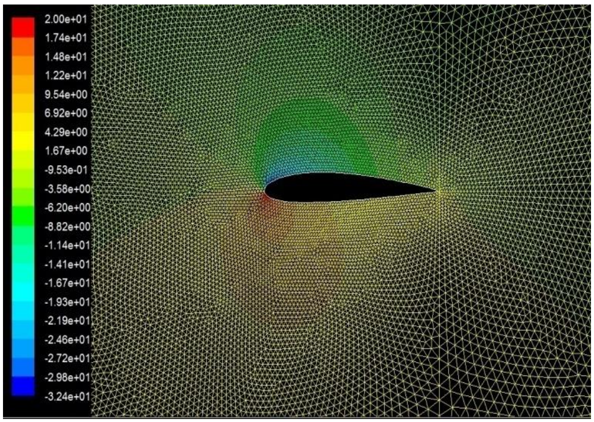

Fig. 18. Contour of Static pressure for α=15^0

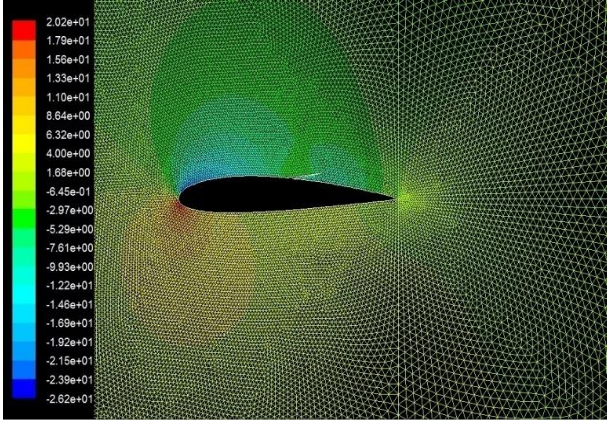

Fig. 19. Contour of static pressure for α=15^0

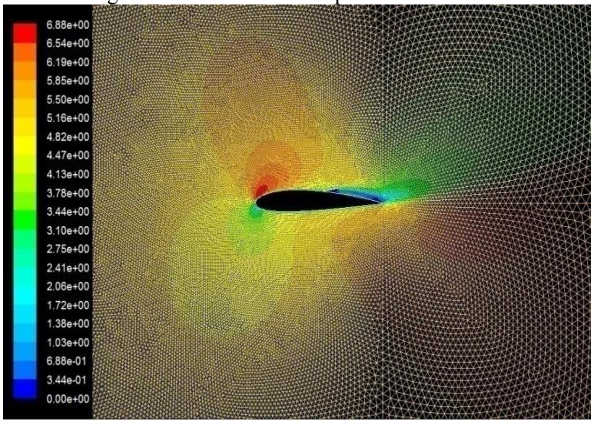

Fig. 20. Contour of velocity magnitude for α=15^0

Contour of the velocity magnitude and static pressure for airfoil without spoiler for angle of attack 15 are shown in the fig. 17 & 18. The effect of the spoiler is shown by the velocity and the static pressure contour at angle of attack 15 in the fig. 19 & 20.They are the indication of the changes that occur during the flow over an airfoil section with spoiler. It is seen that the air flow behind the spoiler is being disturbed. As the angle of attack increases the region of the negative pressure tends toward the leading edge and the pressure difference is decreased. Thus the lift force for the airfoil without any spoiler is higher than that of the airfoil having a spoiler. Also the spoiler resists the flow of the air passing the airfoil, which causes the drag force to be increased.

II. CONCLUSION

This paper demonstrates the effect of spoiler on the aerodynamic characteristics thus the drag and lift force of an airfoil through CFD. The results obtained were quite satisfactory as the main functions of the spoilers were demonstrated clearly through the results. The lift coefficient for the airfoil having spoiler at any of the five positions is lower than that of the airfoil without spoiler. Simultaneously the drag force obtained with a spoiler fitted to the airfoil is higher than that of the airfoil without any spoiler. The purpose of this paper was to observe the effect of the spoiler, with a numerical approach, which ended with a reliable output.

ACKNOWLEDGEMENT

The work has been performed in the Department of Mechanical Engineering, Khulna University of Engineering and Technology (KUET), Bangladesh, under the supervision of professor Dr. Mohammad Mashud, Head, Department of mechanical Engineering. Thanks are given to him for magnanimous guidance and valuable counsel in execution and completion of the work.

REFERENCES

[1] Airfoil - CFD-Wiki, the free CFD reference.
[2] Spoiler - CFD-Wiki, the free CFD reference
[3] Standard k-epsilon model -CFD-Wiki, the free CFD reference.
[4] Weltner, Klaus, Ingelman-Sundberg, Martin - Physics of Flight – reviewed.
[5] NASA Glenn Research Center. Archived from the original on 5 July 2011. Retrieved 2011-06-29.Babinsky, Holger (November 2003).
[6] Halliday, David, Resnich , Robert - "How do wings work".
[7] John Wiley & Sons - "Fundamentals of Physics" - 3rd Edition pp-378.
[8] Vernon H. Gray – "Prediction of aerodynamic penalties caused by ice formation on various airfoils" - Ohio National Aerodynamics and Space Administration Research Center Cleveland.
[9] Santiago Giraldo, Manuel J. Garcia, Pierre Boulanger - "CFD Based Wing Shape Optimization Through Gradient-Based Method " EAFIT University, No 7 Sur 50, Medellin, Colombia University Of Alberta, 2-21 Athabasca Hall, T6G 2E8, Canada.
[10] Mack M.D, Seetharam H.C, Kuhn W.G, Bright J.T. - Aerodynamics of spoiler control devices -- AIAA Aircraft System and Technology Meeting Proceedings, Paper 79 1873, August 1973.
[11] R.A. Stuckey - A Comparison of Spoiler Aerodynamic Characteristics as Estimated from Flight - January 15, 1997.
[12] James E. Fitzpatrick and G. Chester Furlong - Effect of Spoiler Type Lateral-Control Devices On The TWBTING Moments of A Wing Of NACA 230-Series Airfoil Sections - Langley Memorial Aeronautical Laboratory, Langley Field, VA.