Proceedings of the
International Conference on Mechanical Engineering and Renewable Energy 2013
(ICMERE2013) 1 - 3 May 2014, Chittagong, Bangladesh

ICMERE2013-PI-034

FLOW SEPARATION CONTROL OVER AN AIRFOIL BY USING CO-FLOW JET

Mohammad Mashud[1,*], Md. Asaduzzaman[2] and Md. Amzad Hossain[3]

[1-3]Department of Mechanical Engineering
Khulna University of Engineering & Technology
KUET, Khulna-9203, Bangladesh.

[1,*]mdmashud@yahoo.com

Abstract- *The purpose of this research is to investigate the flow characteristics over an airfoil to control the flow separation control of co-flow jet (CFJ) airfoil by CFD simulation. High lift at high angle of attack, increase stall margin and reduction of pressure drag are some performance investigation result for CFJ airfoil. The concept of CFJ airfoil is to open an injection slot near the leading edge from which a high velocity jet is injected tangentially in the same direction of the main flow, as a result energy transferred from the jet to main flow and allow the main flow to overcome adverse pressure gradient, and the flow is remain attached by the help of a suction slot near the trailing edge which suck same amount of air that is injected from the injection slot. For the numerical performance study over NACA2415 airfoil full turbulent boundary layer assumption is used and solution is carried on the basis of 2-D compressible Navier-Stokes equations with 2-equation standard k-ε turbulent model and standard wall function. From simulation result it is seen that the CFJ airfoil delay flow separation over the airfoil and allows the aircraft to cruise with very high aerodynamic efficiency and also enhance the performance of taking off and landing within short distance.*

Keywords: Flow Separation, Airfoil, Co-Flow Jet

1. INTRODUCTION

Several problems involving the flow of fluid around submerged objects are encounter in the various engineering fields in which flow over an airplane is one of the most important. It is due to the fact that when air flow over the airplane wing produced lift determines the performance of the airplane. But for the high speed flow of air over a large solid surface it experience an adverse pressure gradient in the direction of the flow which leads to separation of the boundary layer over the solid surface. Separation of the boundary layer greatly affects the flow as a whole. The disturbed fluid on the downstream form a wake zone which gives rise to boundary forces. As a result lift reduced on the other hand drag increased dramatically. For this adverse phenomenon and to achieve higher aerodynamic performance of the aircraft great effort is done to control this flow separation. Various techniques are proposed to augment flow control and enhance lift including rotating cylinder at leading and trailing edge[1], circulation control using tangential blowing at leading edge and trailing edge[2], providing a partial bumpy on the upper surface[3]. When a flow control technique is developed, there may be three issues needed to be considered: 1) Effectiveness: the flow control method should have substantial improvement of aerodynamic performance, which primarily includes lift enhancement, drag reduction, and stall margin increase (suppression of separation) ; 2)Energy efficient: the flow control method should not cause significantly more energy expenditure. Otherwise, the penalty may outweigh the benefit for the whole aircraft as a system.
This includes minimal penalty to the propulsion system, minimal weight increase due to the FC system; 3) Easy implementation: the FC technique should not be too difficult to be implemented. In this paper CFD analysis is done on a flow control technique over an airfoil which is first proposed and investigated by Ge-Cheng Zha and Craig Paxton in University of Miami which is known as co-flow jet(CFJ) airfoil concept[4]. By this concept a long slot on the airfoil suction surface from near leading edge to near trailing is to open in which a high energy jet is injected near the leading edge tangentially and the same amount of mass flow is sucked away near the trilling edge which causes a strong turbulence diffusion and mixing between the main flow and jet, which enhance the lateral transport of energy and allow the main flow to overcome the severe adverse pressure gradient and stay attached at high angle of attack(AOA)[4]. In case of CFJ airfoil the mass flow rate in injection and suction is same as a result the overall energy expenditure is reduced compare to the separation control by only blowing. Another energy saving of this

flow control method is that it is desirable to blow the jet near leading edge where the pressure is low and to suck the jet near the trailing edge where the pressure is high. In this method no moving parts are needed as a result it's easy to implement. For satisfying these performances 2D CFD analysis is done on NACA 2415 airfoil in this paper.

2. CO-FLOW JET AIRFOIL NOMENCLATURE

Fig.1 shows the baseline airfoil, NACA2415, and fig.2 shows the modification of baseline airfoil to co-flow jet airfoil.The co-flow jet airfoil is modified from the baseline airfoil by translating the suction surface vertically lower. The slot surface shape is exactly the same as the original baseline airfoil suction surface. In co-flow jet airfoil the injection slot is at the leading edge and suction slot at the telling edge. The co-flow jet airfoils are defined using the following convention: CFJ4dig-INJ-SUC, where 4dig is the same as NACA4 digit convention, INJ is replaced by the percentage of the injection slot size to the chord length and SUC is replaced by the percentage of the suction slot size to the chord length. For example, the CFJ2415-167-167 airfoil has an injection slot height of 1.67% of the chord and a suction slot height of 1.67% of the chord. The suction surface shape is a downward translation of the portion of the original suction surface between the injection and suction slot. The injection and suction slot are located at 6.72% and 88.72% of the chord from the leading edge. The slot faces are normal to the suction surface to make the jet tangential to the main flow.

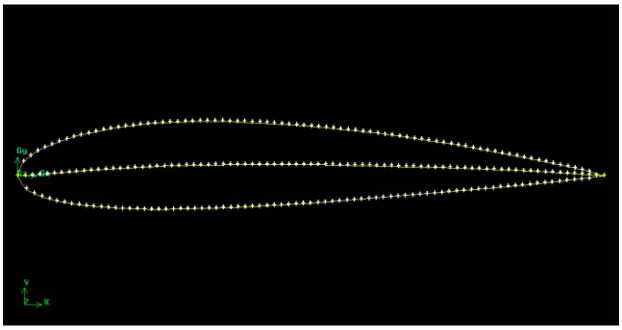

Fig.1: NACA 2415 airfoil

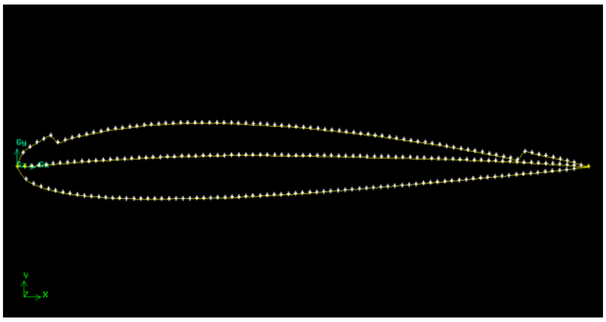

Fig. 2: CFJ 2415-167-167 airfoil.

3. CFD SOLVER

The fluent CFD software is used as the tool to simulate the flow over the co-flow jet airfoil. The mean governing equations are the 2D compressible Navier-Stokes equations. A parabolic farfield boundary is used with the downstream boundary extended to 20 chord length and upstream to 15 chord length. The two equation k-ε turbulence model is used. The k- ε model is selected due to its capability of taking into account of turbulent boundary layer history effect by solving the complete transport equations of *k* and ε. The k-ε model is more capable than algebraic models to predict the separated flows, which occur when the airfoil stalls at high AoA. As the solver does not have any transition model the full turbulent boundary layer assumption is used. The freestream Mach number is 0.3 and the Reynolds number is 3.0×10^6. Very fine structured mesh is used. In the injection slot the jet inlet hold a constant pressure equal to 1.315 times atmospheric pressure and in the suction slot the static pressure is iterated to match the jet inlet mass flow rate.

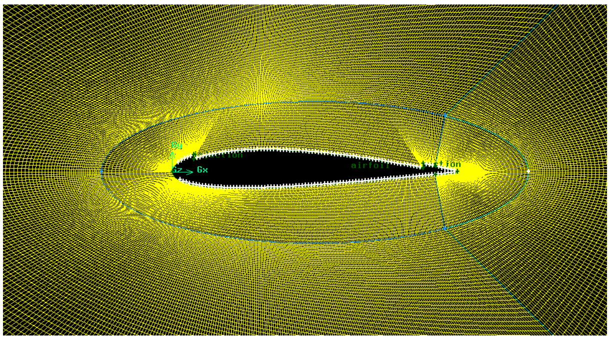

Fig.3: Mesh around the co-flow jet airfoil.

4. RESULT AND DISCUSSION

In figure 4 the simulated data for Re=3.0×10^6 for c0-flow and baseline airfoil is compared. From this figure it is seen that the lift of the co-flow jet airfoil increased dramatically with increased angle of attack and the stall angle is increased by 5˚ where the lift coefficient for co-flow jet airfoil is 2.677 which is 88.52% higher then the maximum lift coefficient for baseline airfoil. As a result the operating range of angle of attack is increased by 38.46%. The zero lift angle of attack for co-flow jet airfoil is around -6˚ and -2˚ for baseline airfoil.

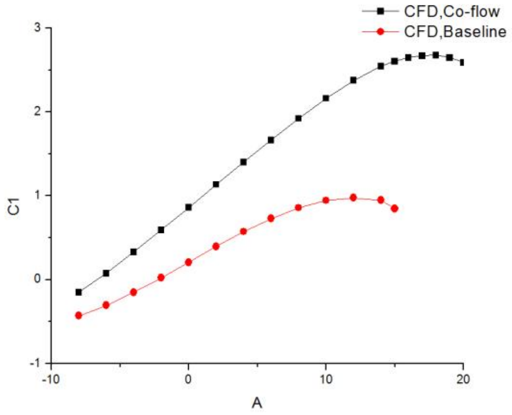

Fig. 4: lift coefficient vs angle of attack.

The Fig. 5 shows that streamlines at AOA =19˚ in co-flow jet airfoil where the flow is smoothly attached to the airfoil surface but the stall angle is 18˚ it is due to the fact that a strong vortex is developed adjacent to this layer which is shown from the velocity contour at AOA=19˚ in fig 7. Where jet energy is mostly used to diffuse the flow to make the flow attached but the provided jet energy is not sufficient to diffuse this large wake hence the pressure drag is larger. In case of baseline airfoil from fig.6 flow is separated even at 14 degree and a strong vortex is generated at the trailing edge of the airfoil which dramatically reduces lift.

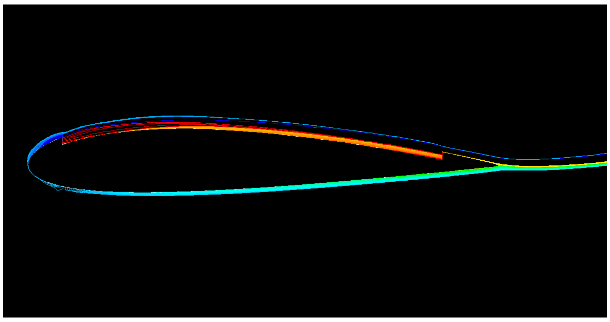

Fig.5: Streamlines at angle of attack of 19 degree for co-flow jet airfoil..

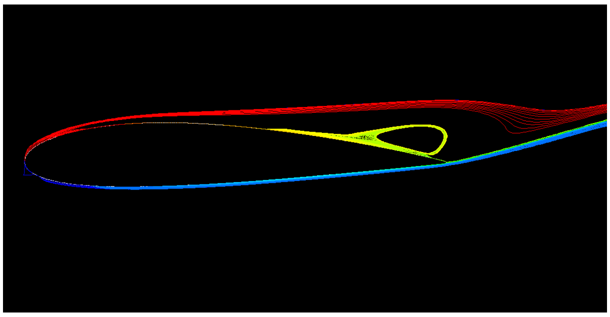

Fig.6: Streamlines at angle of attack of 14 degree for baseline airfoil.

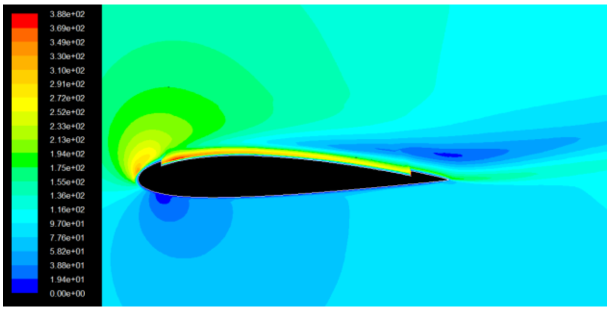

Fig.7: Velocity contour for co-flow jet airfoil at AOA=19˚.

The drag of an airfoil is contributed from two sources, friction drag and pressure drag(form drag).The friction drag will always be in the opposite direction of the flight, that is, always positive. The negative drag hence must be from the pressure dragFig.8 shows that the coefficient of drag for co-flow jet airfoil is larger than baseline airfoil and it increases with increasing AOA. But from the fig. 9 it is seen that the cl/cd is much larger. This increased drag results from the turbulent mixing of jet and main flow. On basis of control volume analysis the drag is determined by

$$D = \iint \rho U (U_\infty - U) dA \qquad (1)$$

Where U is the wake velocity.

When the AOA is very large the jet energy is not sufficient to fill the wake completely, only a part of wake is diffuse with main flow and remain attach with the airfoil, but the remaining wake produce large drag at high AOA. But as the cl/cd is large enough this high drag effect cannot much affect the cruising fight. From fig.9 it is also seen that the maximum cl / cd is obtained at the operating range of 4 to 6 degree for both baseline and co-flow jet airfoil. For this reason the optimum operating range for normal flight is 4 to 6 degree.

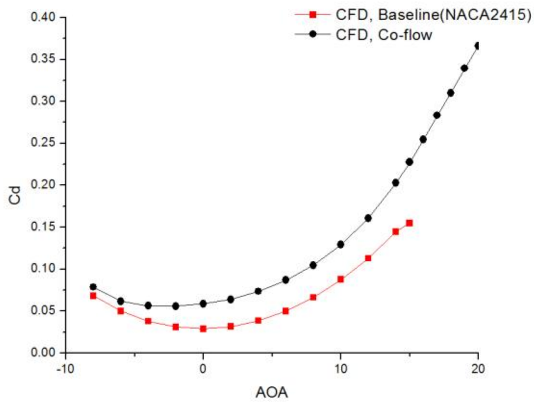

Fig.8: Coefficient of drag vs AOA .

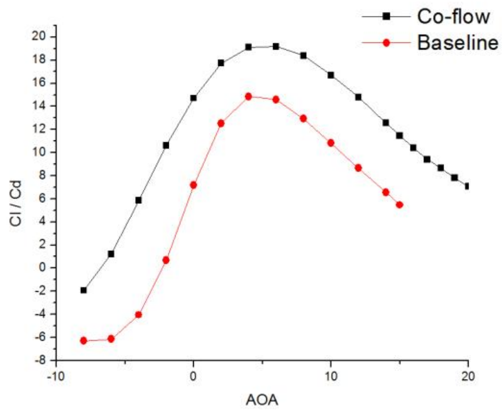

Fig.9: Cl / Cd Vs. AOA.

Fig. 10 indicates that the local maximum Mach number in the suction surface of the co-flow jet airfoil is supersonic and thus creates strong suction effect on the upper surface which is the reason for such high lift. the reason behind this high Mach number is that the high velocity jet will transfer the kinetic energy to the main flow through a low momentum wake. From fig,12 it is seen that the negative pressure region in the upper surface of the co-flow jet airfoil is large. But in case of baseline airfoil from fig.11 it is identified that the local mach number is below the sonic value at 4 degree AOA and this subsonic airfoil is designed and operate in a manner that the maximum local Mach number on the surface of the airfoil must be less then sonic value otherwise it produce strong shock wave on the airfoil surface. Shock wave not generated in the co-flow jet airfoil because it energized the flow by injecting pressurized air.

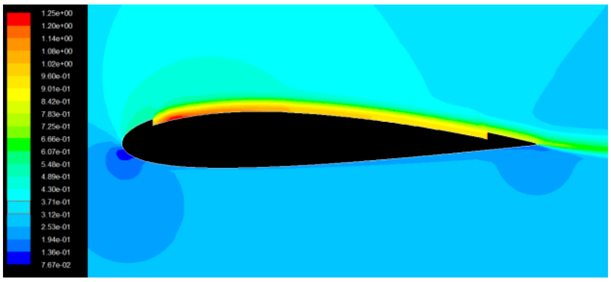

Fig.10: Mach number contours at AOA=4˚ for co-flow jet airfoil.

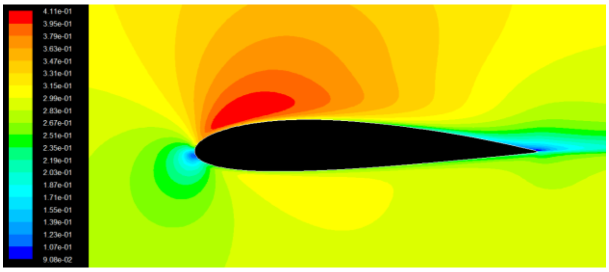

Fig.11: Mach number contours at AOA=4˚ for baseline airfoil.

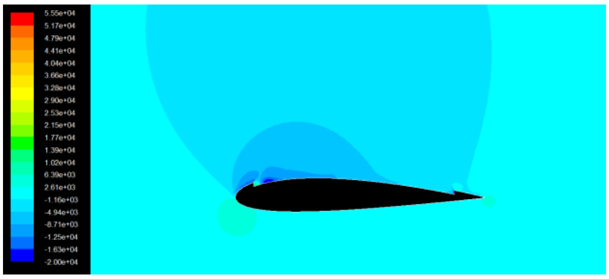

Fig.12:Pressure contours at AOA=4˚ for co-flow jet airfoil.

From fig.13 it is seen that for baseline airfoil wake start to generate even at 8 degree angle of attack. This wake starts to grow stronger with increasing angle of attack and separated the main flow from the airfoil surface. From fig.14, 15 and 16 it can be observed that at the trailing edge of the co-flow jet airfoil at AOA =10˚ no wake is formed but with increasing angle of attack wake is initiate to form at AOA=12˚ because the jet energy is not sufficient to fill the complete weak . The jet mixing only partially fill the wake which make the flow remain attached to the airfoil surface. But with increasing AOA the strength of the wake is also increased dramatically which cause the increasing drag at high AOA. After stall AOA attack such as at 19˚it is observed that the flow is still remain attached to the airfoil(fig.5) but the lift decreased due to the presence of strong wake which produce adverse pressure gradient to the main flow and reduce the suction pressure on the suction surface of the airfoil. It may be possible to delay the formation of the wake on the airfoil suction surface by providing more energy to the jet which may energized the main flow and diffuse the wake. But in this case the energy expenditure may also increase. Experiment may be carried on to evaluate the optimum jet injection pressure for highest efficiency of the airfoil with minimum energy expenditure.

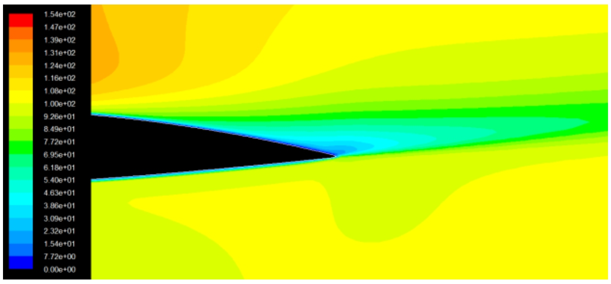

Fig.13: velocity conture at AOA= 8˚ for baseline airfoil.

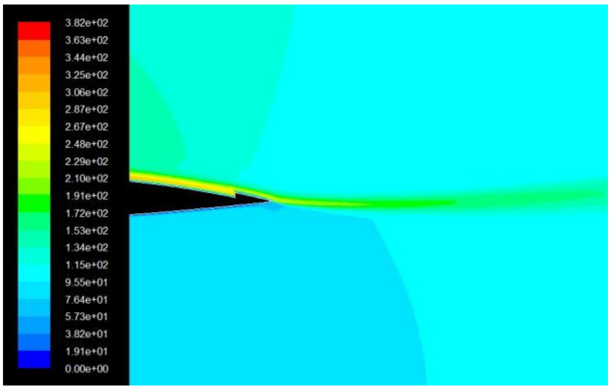

Fig.14:velocity conture at AOA=10˚ for co-flow jet airfoil.

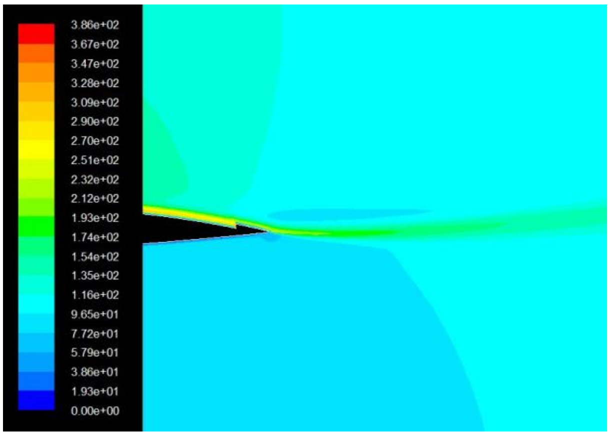

Fig. 15: velocity conture at AOA=12˚ for co-flow jet airfoil.

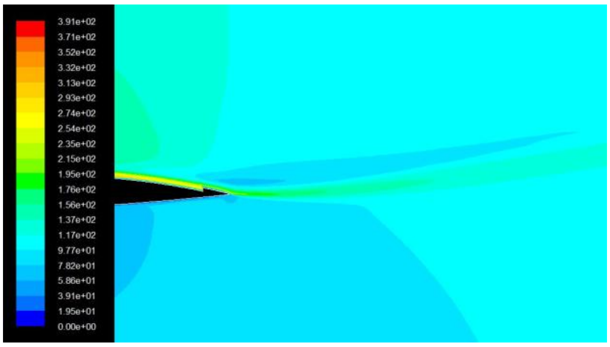

Fig.16: velocity conture at AOA= 15˚ for co-flow jet airfoil.

The new flow control method suggested in this paper appears to have the following advantages: 1) It is an effective method to enhance lift and suppress separation. It can achieve extremely high Cl/Cd at low AOA(cruise), and very high lift and drag at high AOA(taking off and landing).; 2) It significantly increases the AOA operating range and stall margin; 3) It is energy efficient for the overall airframe-propulsion system; 4) It has little geometric limitation and generally can be applied to any airfoil, thick or thin. Compared with the CC airfoil the recirculating CFJ airfoil will significantly save fuel consumption because: 1) the power required to energize the jet is less; 2) no penalty to the jet engine thrust and efficiency since the jet mass flow is not disposed. Above advantages of the CFJ airfoil may derive the following superior aircraft performance:1) The CFJ airfoil works for the whole flying mission instead of only taking off and landing; 2) Economic fuel consumption; 3) Short distance taking off and landing; 4) No moving parts are needed and the implementation is not difficult; 5) Small wing span for easy storage, light weight and reduced skin friction; 6) Low noise since no high lift flap system is used; 7) The CFJ airfoil can be used for low and high speed aircraft.

5.CONCLUSIONS

Co-flow jet airfoil shows revolutionary aerodynamic performance then baseline airfoil which is provided numerically by CFD simulation. This technique can be easily implemented and can be combined with other flow control methods. It enhances lift and delay separation and can achieve dramatically high cl/cd. This significantly increases the AOA operating range and stall margin. In this process energy expenditure is also low. These performances make it capable to use in high and low speed aircraft.

6. REFERENCES

[1] V. Modi, M. Fernando, and T. Yokomizo, “Drag Reduction of Bluff Bodies Through Moving Surface Boundary Layer Control." AIAA Paper No. 90-0298, 1990

[2] N. Wood and L. Robert, “Control of Vortical Lift on Delta Wings by Tangential Leading-EdgeBlowing," Journal of Aircraft, vol. 25, pp. 236-243, 1988.

[3] Md. Abdullah Al Bari, Mohammad Mashud and Hasan Ali, “ROLE OF PARTIALLY BUMPY SURFACE TO CONTROL THE FLOW SEPARATION OF AN AIRFOIL,” ARPN Journal of Engineering and Applied Sciences, VOL. 7, NO. 5, MAY 2012.

[4] G.-C. Zha and C. Paxton, “A Novel Airfoil Circulation Augment Flow Control Method Using Co-Flow Jet." NASA/ONR 2004 Circulation Control Workshop, 16-17 March 2004. AIAA Paper 2004-2208, June 2004.

[5] J. Anderson, J.D., Introduction to Flight, 4th edition. McGraw-Hill Higher Education, 2000.