International Journal of Recent Development in Engineering and Technology
Website: www.ijrdet.com (ISSN 2347-6435(Online) Volume 5, Issue 8, August 2016)

Analysis of the Effect of Moving Surface on 3-D Symmetric Airfoil

Jerrin Varghese[1], Ibrahim Mohammadi[2]
[1]*Lecturer, Department of Aerospace Engineering, Amity University, Dubai Campus*
[2]*Undergraduate Student, Department of Aerospace Engineering, Amity University, Dubai Campus*

***Abstract*— Since the introduction of boundary layer theory by Prandtl, it has always been a great task to control the negative features and always use it for our advantage. Researchers have come up with various methods in controlling the boundary-layer. The use of moving wall and cooling for boundary-layer control has received relatively lesser attention. Irrespective of the method used, the main objective of a control procedure is to prevent or at least delay the separation of the boundary layer from the wall. Since boundary-layer is formed primarily due to friction between fluid flow and stationary surface, conversion of a portion of an airfoil into an endless belt between two rollers could reduce the friction to quite an extent. The highlight in this method is that it injects momentum in the boundary layer and hence preventing separation of flow. The CFD analysis concludes that drag reduction is possible by mounting a number rotating cylinders on the upper surface of the wing. The higher amount of drag reduction is of significant advantage as it reduces the fuel consumption. The concept airfoil showed increase in C_{Lmax} by around 8.8%.**

***Keywords*— Boundary-Layer Theory, Flow Control, Boundary Layer Control, CFD Analysis, Drag Reduction comma.**

I. INTRODUCTION

To artificially control the behavior of boundary layer by analyzing the effect of moving surface and cooling on the lift of a 3-D symmetric airfoil. A stationary cylinder does not produce any lift, but a rotating cylinder produces lift. This is very well explained by Magnus effect which states that, on a rapidly spinning cylinder, there is always an upward force that is applied on the cylinder which is right angled to the axis of spin. Similarly, a symmetric airfoil does not produce any lift at zero degree angle of attack, but theoretically it is said that if a moving belt is attached on an airfoil, it will produce a lift at zero degree angle of attack. This helps in reducing the drag formed and on the contrast, increases the lift produced on the wings. The injection of the air into the already formed boundary layer cause an imbalance in the wing which helps reduce the heat produced. This project will be testing this theory in experiment by various methods such as CFD and flight testing.

A. Boundary Layer

It is a region in which fluid in which the velocity of the flowing fluid changes from 0 on the surface to a definite value ‘U’ at a constant distance from the surface. The fluid flows parallel to the defined material as if the material was absent. Boundary Layer Separation is a phenomenon in which the thickness of the boundary layer increases in the downward direction. Due to this, the velocity decreases, leading to the creation of vortices, subsequently causing massive loss of energy. The main reason for the formation of boundary layer would be the difference in pressure [6].

Formation of Boundary-Layer on an aerofoil shaped body leads to the following drawbacks

- Increase in the Displacement thickness hence increasing the pressure difference.
- Increases pressure drag over the shaped body under flow.
- Formation of shedding vortices [7].

B. Boundary Layer Control

There are several methods which have been developed for the purpose of artificially controlling the behavior of boundary layer. High lift due to reduced drag is the most desired property in the field of aerospace. This can be obtained by the following methods.

Motion of solid wall: This is the most common method for avoiding separation of boundary layer. If we can permit the motion of solid wall along with the velocity of the fluid, it is possible to eliminate the formation of boundary layer because the difference of velocity of the fluid and the solid wall is the main cause for the formation of boundary layer.

The easiest way of achieving this is to rotate the cylinder placed in motion of the fluid at right angle to its axis. Separation is completely eliminated on the upper side where the flow and cylinder moves in the direction. However on the lower side, where the direction of flow of fluid is opposite to that of the solid wall, separation is developed incompletely [6].

Acceleration of Boundary Layer (Blowing): Another method to prevent separation is to supply additional energy to the fluid particles. This can be done by discharging fluid from interior of the body with the aid of a blower or by attaining the required energy directly from the main stream. This is produced by giving higher pressure through a slot in the wing [6].

In both the cases, additional energy is provided to the particles of the fluid in the boundary layer near the wall. They are carefully designed so that shape of the slit does not cause vortices to be formed. Further experiments; have shown blowing at the latter end leads to be better and maximum lift. In cases of slotted wings, the boundary layer formed by forward slat is carried to main stream before separation occurs and from forward of the flap onwards a new boundary layer is created. Under these new conditions, the boundary layer will reach the trailing edge of the aerofoil without any separation. This phenomenon of slot formed in trailing edge by flap is same as principal to that formed at the forward slat.

Suction: The effect of suction is to remove the de – accelerated fluid particles from boundary layer before they are separated; with suitable slit arrangements and under favorable conditions separations can be prevented completely. Simultaneously amount of drag is also reduced due to absence of separation. Due to suction, greater pressure is obtained on the top of airfoil leading to larger maximum lift.

In suction, it also helps in reducing the drag. By using suitable arrangements, it is possible to shift the point of transition of boundary layer from laminar to turbulent. This causes lesser drag because drag in laminar flow is much lesser than the drags in turbulent flow. This is due to the reduction in the size of boundary layer which then becomes less prone to turn turbulent [6].

Cooling of Wall: Studies show that at a certain range of supersonic Mach Number, it is possible to stabilize the boundary layer by cooling the wall of the surface. Cooling can be applied to reduce the thickness of boundary layer hence providing more lift and less drag to the flow [6].

II. BACKGROUND STUDY

The Department of Aeronautical Engineering of the Military Institute of Science & Technology conducted many experiments with the use of a concept NACA 0010 airfoil on the aerodynamics of a wing with a rotary cylinder at leading edge of the wing. CFD was performed by converting the 2-D NACA airfoil into a 3-D aircraft wing.

The cylinder was made to rotate in anti-clockwise direction at the leading edge at first a fixed rpm, and various angle of attack and then at a fixed angle of attack and rotated at various rpm. Giving anti-clockwise rotation, above the cylinder moves the point of separation downwards making the flow above the surface larger than that below the cylinder, and hence making velocity at surface above higher the surface below.

The research was based on Bernoulli's equation (1) that at any time in a flow region, the static and dynamic pressure is always a constant.

$$P + \frac{1}{2}\rho V^2 + \rho g h = Constant \qquad \text{\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_ (1)}$$

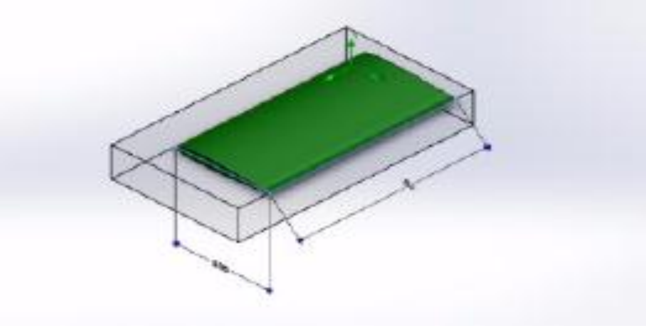

Figure 1. Model NACA 0010 with Leading Cylinder [1]

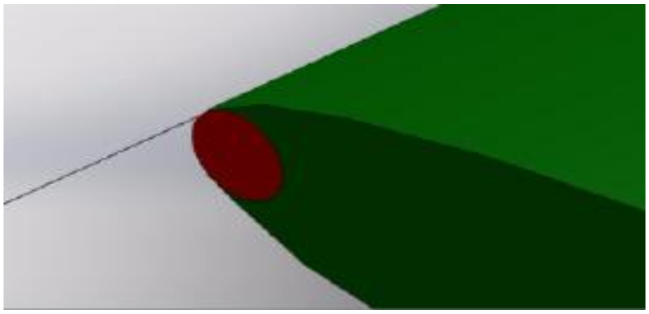

Figure 2. Magnified View of Rotary Cylinder at the Leading Edge [1]

The design challenge was to incorporate the rotating cylinder in a NACA 0010 airfoil, without having to change its shape as well as not to change the chord length of airfoil, and yet have the flexibility to rotate the cylinder. After, it was designed in 2-D, it was extruded to give a 3-D shape. The scale was based 1:100 with a wing span of 0.10m. A rectangular wind tunnel was made around the wing for flow simulation. Another, normal NACA 0010 was also created with same chord and wing span for comparison purposes and to analyze the results [1].

TABLE I.
SIMULATION PARAMETERS USED DURING THE STUDY [1]

Pressure	101325 pa
Temperature	288 K
Velocity	0.60 Mach
Rotation speed	3500 rpm

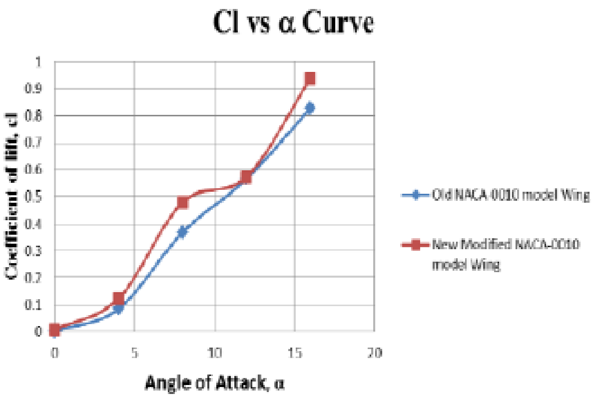

Figure 3. C_L Vs AOA for both types of NACA 0010 [1]

The studies showed a significant impact on the normal NACA 0010 airfoil by the addition of a rotating cylinder at the leading edge is as follows:

- Lift increase is possible by addition of rotating cylinder. Drag also has increased due to additional resistance.
- The modified airfoil showed increase in $C_{L(Max)}$ by around 13.25%.
- The higher amount of drag is of little disadvantages when compared to high benefits of increase in lift [1].

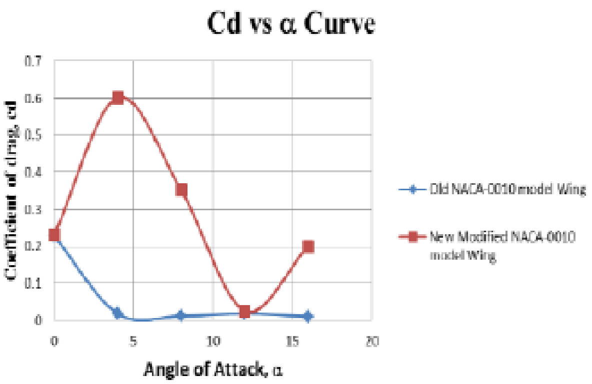

Figure 4. C_D Vs AOA for both types of NACA 0010

Oran University of Science and Technology conducted a study on the numerical analysis on the control of flow separation with respect to a NACA 63218 airfoil by using a moving surface. First, a non-modified NACA 63218 airfoil study is performed for comparing the study of the modified NACA 63218 airfoil at different angles of attack, at a set rpm of the moving surface. The cylinder was place for rotation at leading edge of the wing. The turbulence is modeled on the basis of two equation of K-epsilon [2].

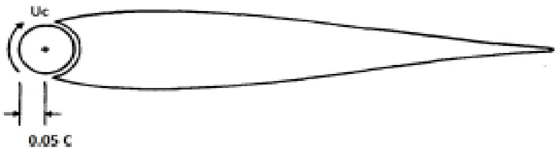

Figure 5. Modified Airfoil NACA 63218 (Chord=0.15m) [2]

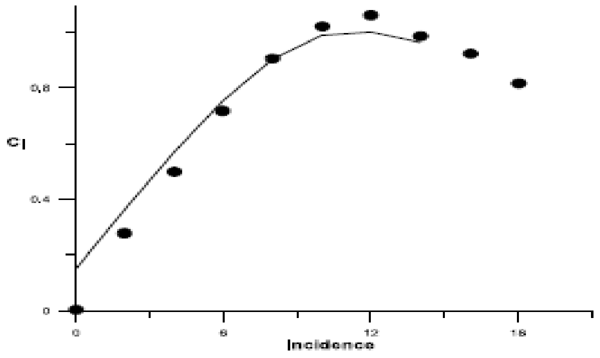

Figure 6. C_L vs AOA for NACA 63218 [2]

In the experiments conducted, structured mesh is constructed. The simulation conducted was run at various angles of attack starting from 0 degree to 32 degree. The Reynolds number was taken as 2.13e+05 and inlet velocity of 20 m/s [2].

The research showed the application of moving surface had several benefit as to when compared to a typical airfoil. The results showed the angular momentum of moving surface increases and subsequently the drag force decreases. In addition to increase in lift, moving surfaces also reduce the effect of stall at high angles of attack [2].

V J Modi from the University of British Columbia describes developments in the field of the moving surface boundary-layer control (MSBC). To begin with, application of this concept was carried out by placing rotating cylinders on the leading edge and / or trailing edge as well as the top surface of the Airfoil [3].

There was a high increase in lift and a delay in stall by applying such a technique. By using appropriate combination of cylinder location and speed, it was found that various levels of performance could be obtained. Results showed that injection of momentum through moving surfaces, achieved here by introduction of bearing-mounted, motor-driven, hollow cylinders, can significantly delay separation of the boundary layer and reduce the pressure drag [3].

In wind tunnel testing, model had 44 pressure taps distributed all over. Chord length being 0.38 m and span being 0.68m. Speed was varied between 1 to 50 m/s and at Reynolds number 4.62e + 04 [3].

It was found that the leading-edge cylinder was quite effective in increasing the lift curve, without considerably changing its slope, thus greatly increasing the maximum lift coefficient and postponing the stall angle. The $C_{L(Max)}$ got with the leading-edge and forward upper-surface cylinder was about three times that of the base combination [3].

When both the cylinders are even with the top face of the trailer, it was clear that the front cylinder rotation reduces the drag coefficient rather considerably, to about a drop in 15.8%. Rotation of the back cylinder improves the situation additionally, and with a combination of both the cylinder the reduction in C_D reaches to amounts up to 22.8% [3].

The effect of raising the rear cylinder drag coefficient is slightly reduced ($C_D = 1.19$). This may be as a result of a combined effect of an increase in the expected area on which the drag coefficient is based and the large wake width caused by the rear cylinder [3].

Various studies have shown about the growing importance of CFD, while replacing the wind tunnel in the future as the technology behind CFD improves and computers become more powerful. At the moment, CFD can give results almost as accurate as a wind tunnel that are often more useful due to the sophisticated visualization and domain wide measurements characteristic of CFD. Though Wind Tunnel Modelling is generally accepted in the scientific and engineering world and is proven to be the representative for real world situations but with CFD, it is a well-proven tool that was economically feasible only on mainframe computers until recent advances in computing made it possible to use a desktop PC. However, CFD results may not be as comprehensively comparable to real world results as most wind tunnel results can be [4].

Advantages of CFD over wind tunnel modelling vary from being cheap, providing a better visualization of result.

Every advantage comes along with its own disadvantage and in this case, CFD has its own set of problems like it can be too erroneous, the projects can't be too complex and it is not an accepted industry standard. By different examples taken and tested on, it was found out that the CFD results gave out almost the same results to that of a wind tunnel and is a reliable accurate tool but it falls short due to its deviation from the real world results [4].

This experiment speaks about the use of serpentine boundary layer ingesting diffusers offers a significant benefit to the performance of a blended wing body aircraft. After conducting various tests, the conclusion was that computational analysis showed heavy reduction in the Pyramid ejector and the circumferential ejector by 71% and 68% respectively. These differences were incurred due to the result of jet interaction. It was also concluded that by keeping the jet flows separate and distinct, the diffuser secondary flows could be managed and for this the most practically effective flow control scheme was the circumferential ejector scheme [5].

III. NUMERICAL DATA ANALYSIS

The simulation was carried out using the following settings. The simulation was carried out at different positive Angle of Attacks up to 20°.

CFD Parameters

1. Density: 1.225 kg/m3
2. Viscosity: 1.7844e-05 kg/m-s
3. Velocity: 44 m/s
4. Reynolds Number: 3,000,000
5. Temperature: 288 K

Grid Quality (Mesh)

1. Mesh Element: Tetrahedron
2. Mesh Type: Patch Independent
3. Maximum Element Size: 68 mm
4. Transition: Fast
5. Relevance: Fine
6. Mesh Skewness : 0.6 < Skewness < 0.85

The aerofoil used for the simulation was NACA 0012 with the following specifications throughout the analysis.

Design Specifications

1. Chord Length: 350mm
2. Wing Span: 2100mm
3. Aspect Ratio: 6
4. Number of Cylinders: 5
5. Cylinder Diameter: 4mm

A. Analysis for Concept Airfoil at 4 degree Angle of Attack @500 rpm

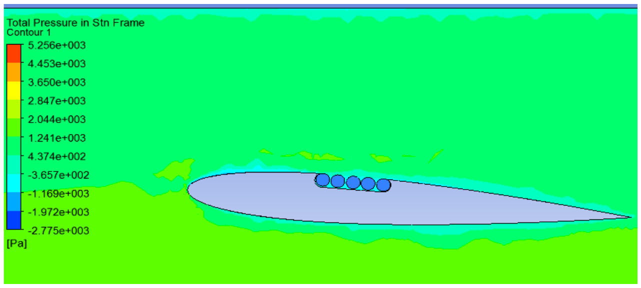

Figure 7. Pressure Contour of concept airfoil at 4 AOA at 500rpm

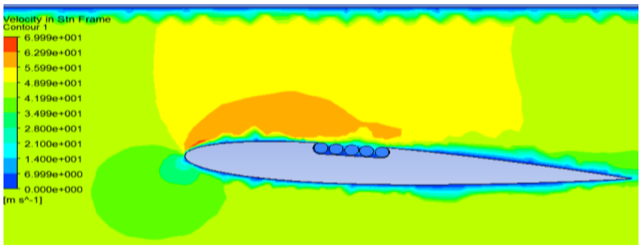

Figure 8. Velocity Contour of concept airfoil at 4 AOA at 500rpm

B. Analysis for Concept Airfoil at 8 degree Angle of Attack @500 rpm

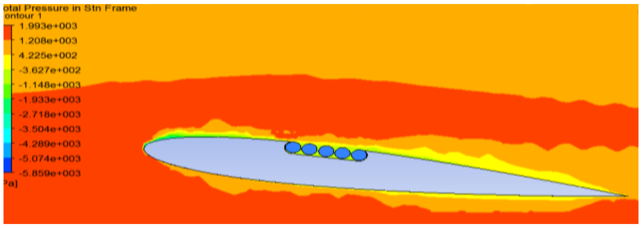

Figure 9. Pressure Contour of concept airfoil at 8 AOA at 500rpm

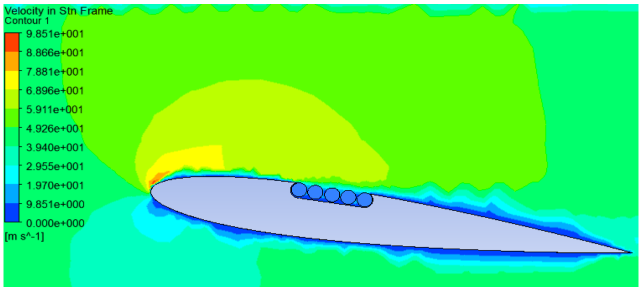

Figure 10. Velocity Contour of Concept Airfoil at 8 AOA at 500rpm

C. Analysis for Concept Airfoil at 12 degree Angle of Attack @500 rpm

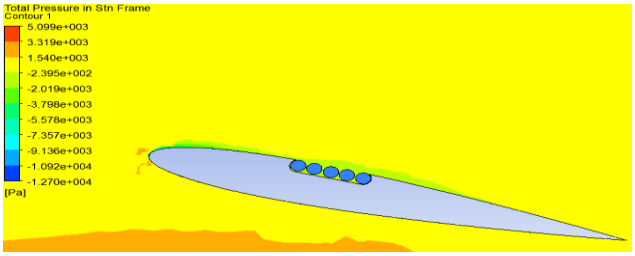

Figure 11. Pressure Contour of concept airfoil at 12 AOA at 500 rpm

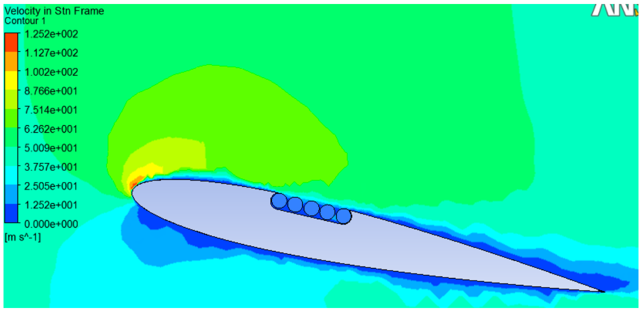

Figure 12. Velocity Contour of concept airfoil at 12 AOA at 500rpm

D. Analysis for Concept Airfoil at 16 degree Angle of Attack @500 rpm

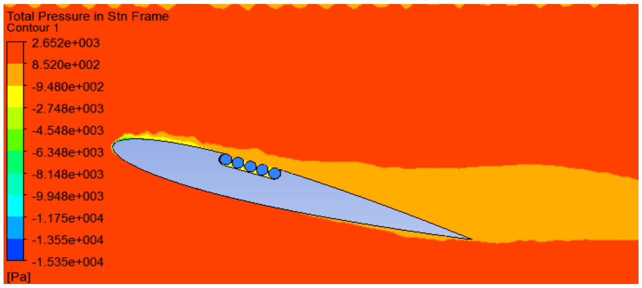

Figure 13. Pressure Contour of concept airfoil at 16 AOA at 500rpm

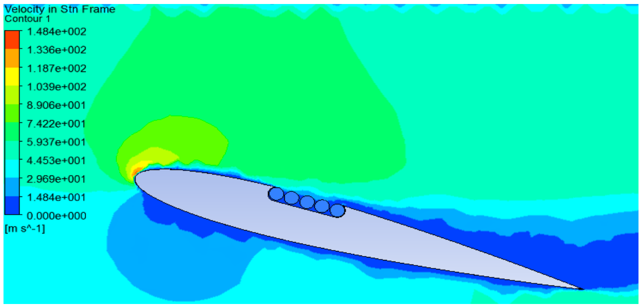

Figure 14. Velocity Contour of concept airfoil at 16 AOA at 500rpm

E. Analysis for Concept Airfoil at 20 degree Angle of Attack @500 rpm

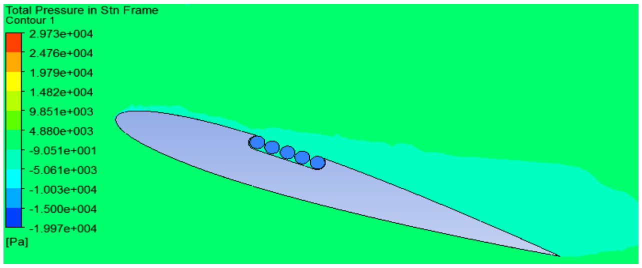

Figure 15. Pressure Contour of concept airfoil at 20 AOA at 500rpm

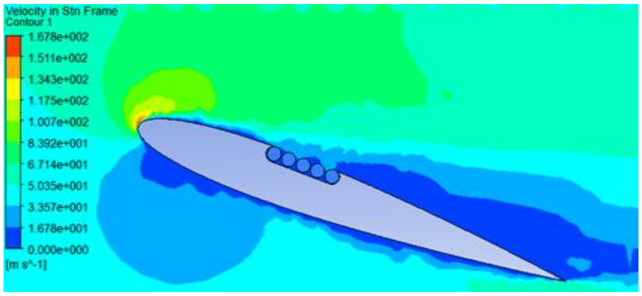

Figure 16. Velocity Contour of concept airfoil at 20 AOA at 500rpm

The below tables and graphs gives the result that are obtained from the computational analysis that shows the effect on lift and drag by addition of moving surface (cylinders) rotating in accordance to the motion of the flow.

TABLE III
COMPARISON OF C_L AND C_D VALUES OF THE CONCEPTUAL AIRFOIL AGAINST THE EXISTING AIRFOIL

AOA	C_L-Concept	C_L-Existing	C_D-Concept	C_L-Existing
0	-0.0414	0.000015	0.0102	0.1386
2	0.1418	0.292	0.0129	0.01601
4	0.3464	0.5035	0.0178	0.02807
6	0.5604	0.7058	0.0233	0.0351
8	0.7656	0.8114	0.0334	0.04191
10	0.9286	1.001	0.0454	0.04739
12	1.1329	1.2304	0.06103	0.07321
14	1.2524	1.3871	0.081	0.08917
16	1.421	1.6688	0.1193	0.1189
18	1.6922	1.5549	0.1601	0.1546
20	1.586	1.388	0.2494	0.2159

TABLE IV
CL AND CD VALUES OF THE CONCEPTUAL AIRFOIL FOR VARYING RPM AT CONSTANT 4° AOA

RPM	Coefficient of Lift(C_l)	Coefficient of Drag(C_D)
0	0.345752	0.018179
1000	0.3485781	0.0177072
2000	0.3489683	0.0179169
3000	0.3468858	0.0181704
4000	0.3484699	0.01788876
5000	0.34902	0.0177669
6000	0.349126	0.0179397
7000	0.3484229	0.0179032
8000	0.3479528	0.0181872

IV. RESULTS AND DISCUSSIONS

The figure below shows the relationship between co-efficient of lift vs. Angle of Attack. It is clear that the existing airfoil has constant higher lift than concept airfoil up until 16 degrees and then it stalls. Concept airfoil has higher co-efficient of lift (1.6922), when compared to existing airfoil's co-efficient of lift (1.6688). Concept airfoil reaches its $C_{L(Max)}$ at higher angle of attack (18 degrees), while compared to existing airfoil (16 degrees), before stall happens. At 18 degrees, the co-efficient of lift increases by about 8.8%.

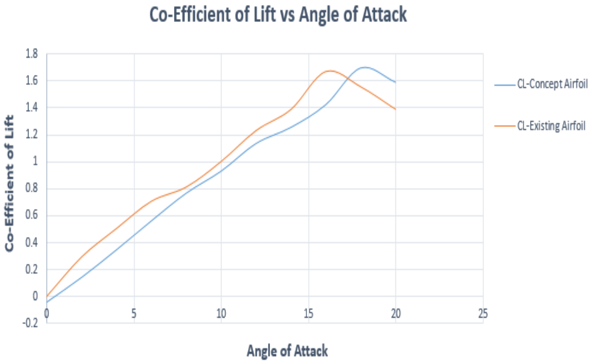

Figure 17. C_L vs. AOA for both types of NACA 0012 wing

The figure below shows the relationship between co-efficient of drag vs. Angle of Attack. It is clear that drag has reduced in concept airfoil as to compare to existing airfoil. Concept airfoil has lower co-efficient of drag till 16 degrees and then it increases significantly, this is mainly due to large number of vortex formed behind the rotating cylinders at large angle of attack. At 4 degrees, co-efficient of drag reduction is about 57.6%. While, at 20 degrees co-efficient of drag increase is 15.5%.

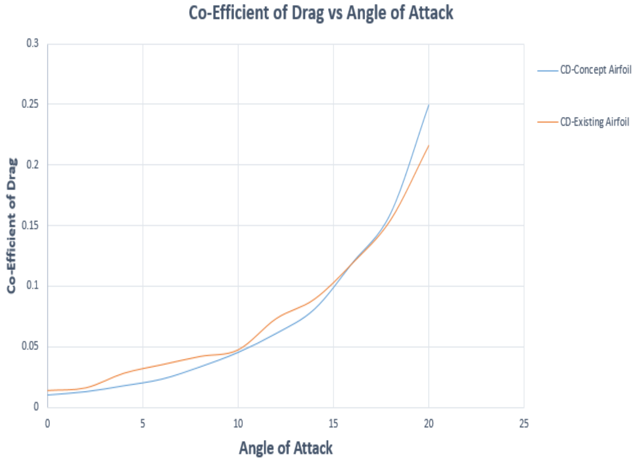

Figure 18. C_D vs. AOA for both types Of NACA 0012 wing

The figure below shows the relationship between co-efficient of drag vs. co-efficient of lift. It is clear from the graph that C_D/C_L ratio for existing airfoil is consistently higher up to co-efficient of lift value of 0.8. Between 0.8 and 1.2 values of co-efficient of lift the ratio is almost same and 1.2 onwards the ratio of C_D/C_L becomes favorable for existing airfoil up until stall. It is also seen that $C_{L(MAX)}$ is higher for concept airfoil and is consistent with the results from previous graph.

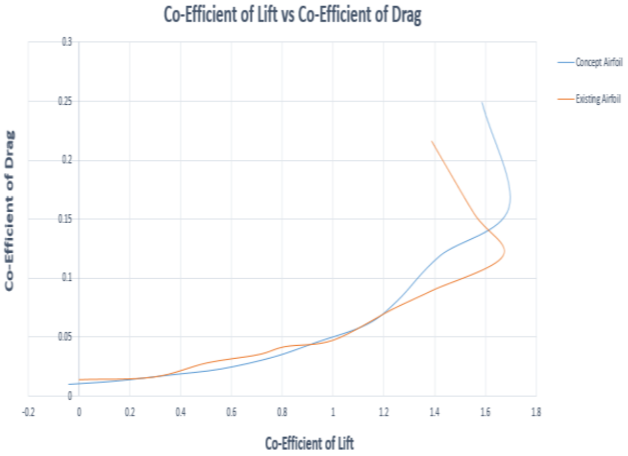

Figure 19 C_L vs. C_D for both types of NACA 0012 wing

The figure below shows the relationship between co-efficient of lift vs. Rotations per minute of the rotating cylinder. Concept airfoil has rotating cylinders which are rotated from 0 rpm up to 8000 rpm with an interval of 1000 rpm at a constant 4 degree angle of attack. It is clear from the graph that the value of co-efficient of lift is lowest at 0 rpm (0.345752). We see a large increase in C_L from 0 to 1000 rpm and then a small increase till 2000 rpm and then see a sudden fall on 3000 rpm. From 3000 rpm, there is again increase in C_L and reaches its maximum value at 6000 rpm (0.349126) and then it falls again on reaching 7000 rpm and 8000 rpm respectively. There is 1% increase in the value of C_L when the lowest value of C_L is compared to the highest value of C_L.

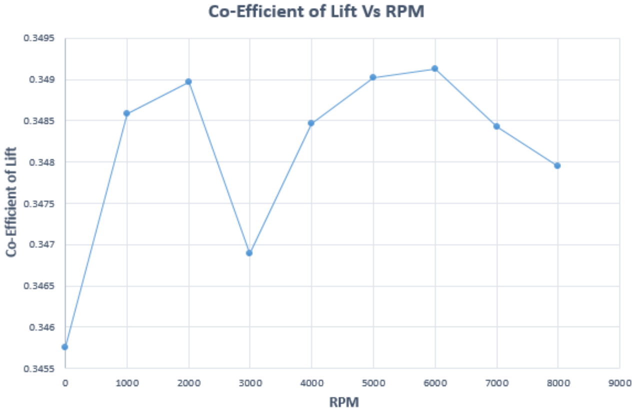

Figure 20. C_L vs. RPM (Rotating Cylinder) for concept airfoil of NACA 0012 wing

The figure below shows the relationship between co-efficient of drag vs. Rotations per minute of the rotating cylinder. It is clear from the graph that the value of co-efficient of drag is lowest at 1000 rpm (0.017702). It is highest at almost three points 0 rpm (0.018179), 3000 rpm (0.018170) and 8000 rpm (0.0181872). We see a large decrease in C_D from 0 to 1000 rpm, reaching its lowest point and then a sharp increase till 3000 rpm and then see a fall up to 5000 rpm. From 5000 rpm, there is increase in C_D till 6000 rpm and then a short decrease till 7000 rpm and then reaching its maximum value of C_D at 8000 rpm. There is 2.75% decrease in the value of C_D when the highest value of C_D is compared to the lowest value of C_D.

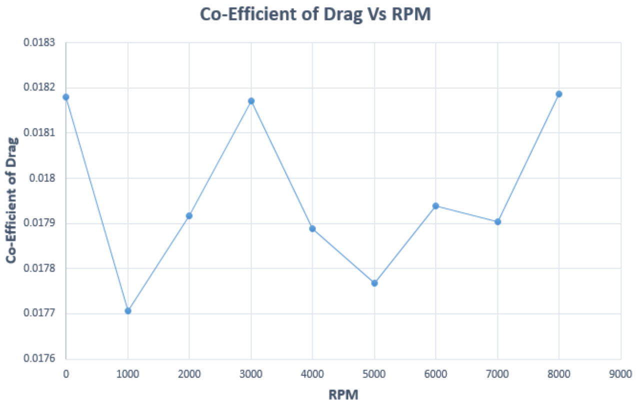

Figure 21. C_D vs. RPM (Rotating Cylinder) for concept airfoil of NACA 0012 wing

V. CONCLUSIONS

By performing Computational Analysis of a modified symmetric 3-D NACA 0012 airfoil with rotating cylinders, we found the following:

1) Drag reduction is possible by mounting a number rotating cylinders on the upper surface of the wing. This data is justified for different configurations also. This is of significant advantage as it reduces the fuel consumption. But, lift is reduced slightly. This may be due to some poor design.
2) The introduction of rotating cylinder extends the lift curve, without drastically changing its slope, thus increasing the maximum lift co-efficient. The concept airfoil showed increase in C_{Lmax} by around 8.8%. This is desirable for takeoff, when runway length is short.
3) Co-efficient of lift has a maximum value of 0.349126 at 6000 rpm and minimum value of 0.345752 at 0 rpm. It shows about an increase of 1%.
4) Co-efficient of drag has a minimum value of 0.017702 at 1000 rpm and maximum value at almost three points 0 rpm (0.018179), 3000 rpm (0.018170) and 8000 rpm (0.0181872). It shows about a decrease of 2.75%.

On the basis of graph 4 and 5, we can come to a conclusion that 5000 rpm is the most desired as it has the best C_L /C_D ratio.

REFERENCES

[1] "Shuvrodeb Barman, Asif Shahriar Nafi, Nafisa Nawal Probha, and Saif Atique . A CFD Based Parametric Study to Investigate the Moving Surface Effect on Airfoil Boundary-Layer Control .MIST, Dhaka, 2015.

[2] Tayeb Yahiaoui, T, 2015. Effect of Moving Surface on NACA 63218 Aerodynamic Performance. EPJ Web of Conferences.

[3] Modi, V.J, 1997. MOVING SURFACE BOUNDARY-LAYER CONTROL: A REVIEW. Journal of Fluids and Structures, Journal of Fluids and Structures (1997) 11, 627-663.

[4] J.D. McAlpine. 2005. Computational Fluid Dynamics or Wind Tunnel Modeling?

[5] Harouni, A.G, 2014. Flow control of a boundary layer ingesting serpentine diffuser via blowing and suction. Aerospace Science and Technology, Volume 39, December 2014, Pages 472–480.

[6] Schlichting, H, 1970. Boundary-Layer Theory, 8th ed,, New York. McGraw-Hill [Accessed 08 February 2016].

[7] Chang, P.K., 1970. Separation of Flow. 1st ed. New York: Pergamon Press [Accessed 08 February 2016].