Numerical Simulation of Flow Over Airfoil and Different Techaniques to Reduce Flow Seperation Along with Basic CFD Model: A Review Study

P. B. Makwana[1], J. J. Makadiya[2]
PG Student, Mechanical Engineering Department, RK University, Rajkot, Gujarat, Indi a [1]
Assistant Professor, Mechanical Engineering Department, RK University, Rajkot, Gujarat, India[2]

***Abstract*--To take of aircraft within short distance and to increase efficiency of turbo-machinery lift force required to increase and drag required to reduce. These mainly depend on the lift coefficient and drag coefficient. When flow pass over airfoil at some angle due to the adverse pressure effect flow is going to separate from surface so it reduce the lift and increase drag. So in this paper mainly focused on the different technique to reduce flow separation and some general idea about different modal of CFD.**

Keywords: Airfoil, flow separation, CFD model

I. INTRODUCTION

Wings is consider as most important part of the aircraft, without wings aircraft can't fly. Science the geometry of the wings has influence on the other aircraft component. This wing is of airfoil section. Whenever flow is passed over airfoil there are basically two types of forces are generated one is lift force and other is drag force.

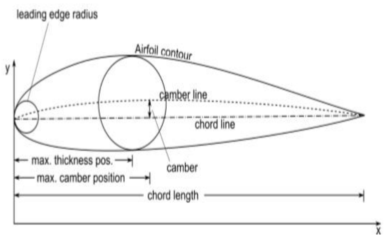


Figure 1 Basic airfoil design and definition parameters

Lift force is the force which is perpendicular to the flow direction and drag is the force which is parallel to flow direction. Drag is depend on the body surface and fluid which is flow over it. If fluid consider as inviscied then the wall shear stress part is not consider. Lift generated by airfoil depends on the air density, velocity, viscosity, surface area, shape of airfoil, angle of attack and if flow is consider as compressible then on compressibility, all this variable characterized by single variable call lift coefficient C_L . How air density, Mach number, and Reynolds number very with height are presented by Rong Ma et al [2009]

$$L=\frac{1}{2}\rho U^2 S C_L$$

Likewise for the drag forc D that single variable like in lift it's C_D drag coefficient

$$D = \frac{1}{2}\rho U^2 S C_D$$

C_L Are depends on the pressure distribution on airfoil. S.kandwal et al [2012] has done the simulation of invicid flow over airfoil and found that the pressure coefficient is maximum at the point of flow attack, lower on the upper surface and velocity is high at upper surface.

Adverse pressure gradient is due to the stream wise pressure force tends to flow to counter the shearing effect and resulting retarded flow nearer the wall as the pressure increase alone the wall pressure gradient become adverse due to flow decelerates

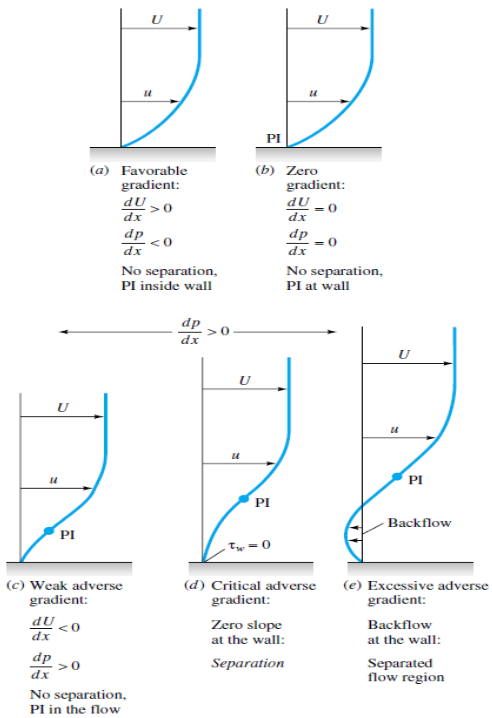


Figure 2 Boundarly layer profiles under different pressure gradients. Effect on point of inflection

Due to a strong adverse pressure gradient effect flow become reverse and flow separated from surface. Flow separation mainly connected with lift capability, by preventing flow separation reduce the total drag to such extend

II. NUMERICAL SIMULATION OF FLOW OVER NACA0012

Hua shan et al [2004] has done the DNS of flow separation around NACA0012, due to the flow separation separated layer is generated which is inviscidly unstable and vortices are generated that is due to Kelvin helmont's instability mechanism this instability is predicted by liner stability [LST]theorem or parabolic stability equation[PSE] but due to the assumption in LST of parallel base flow and in PSE of steady flow so out of these can't be use involving flow separation .

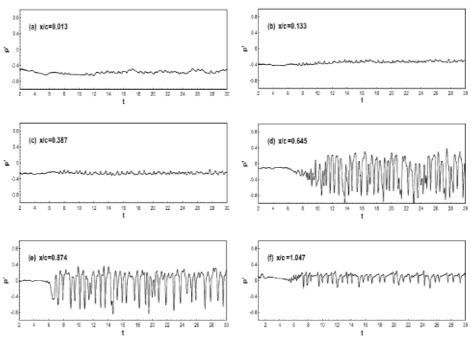
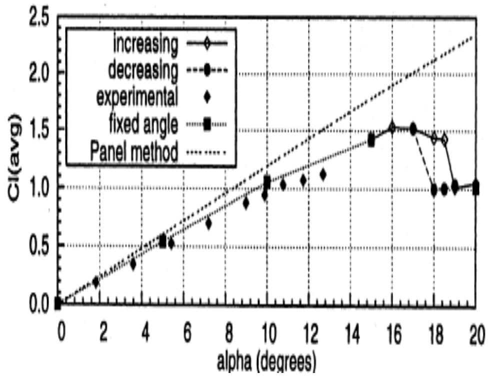


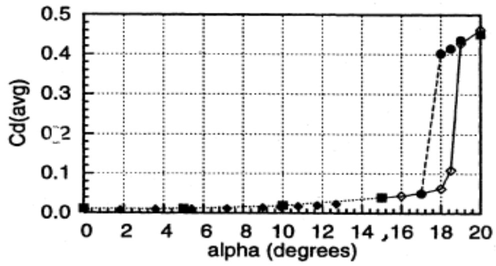
Figure 3 Time history of pressure fluctuation at different locations

Above figure is at the same time step pressure fluctuation at various chord location from that it's clear that pressure fluctuation first start at the trailing edge, that is due to the counter direction of flow of upper and lower surface at the trailing edge and due to that weak are generated. In fig. (d) and (f) time lag indicated that the disturbance are propagated in the form of acoustic wave.

Sanjay Mittal et al [2002] has done Hysteresis in flow past a NACA 0012 airfoil with increasing and decreasing angle of attack, they observed by increasing angle of attack flow separation leads to wards leading edge. Figure show the variation in C_L and C_D by increasing and decreasing angle of attack, by increasing angle of attack unsteadiness is less compare to decreasing angle of attack they also observed that by increasing angle of attack stall occur at 19° while decreasing stall occur at 17°



(a)



(b)

Figure 4 (a) and (b) Re = 10^6 turbulent flow past a NACA 0012 airfoil: variation of the time-averaged drag and lift coefficients r with angle of attack

III. DIFFERENT TECHNIQUE TO REDUCE FLOW SEPARATION

A. *Suction effect on boundary layer*

The idea of boundary layer flow control introduce by L.Prandtal. They focus on the boundary layer suction, on progress of research they investigate on both suction and blowing.

M.Goodarzi et al [2012] has done suction effect on NACA0012, suction effect is investigated by changing two parameter slot location on upper side and for each slot three different suction ratio. Result show that slot location with 10% chord length and suction ration of 0.5 has great effect on the lift and drag coefficient and also stall condition change from 14° to 20°.

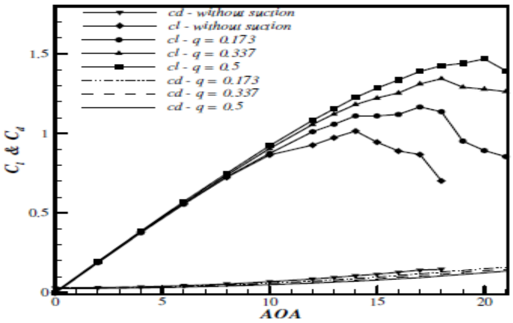


Figure 5 Lift and drag coefficients of the NACA 0012 at *Re* = 5×10^5 and slot location of 10% of the chord length.

Apostolos et al [2010] Has done work on the suction effect on the NACA4412, they observed the at the 0° angle of attack if we applied suction at minimum pressure point and suction effect normal to the wall direction then it's only applicable to the particular angle of attack, also they change the suction at 45° so no effect on the transition point but if suction is applied at the leading edge has great effect for both small as well as high angle of attack. They also compare result of discrete versus distributed suction and found that distributed suction has good effect on transition and suction coefficient value of 0.08 beyond the no more noticeable effect.

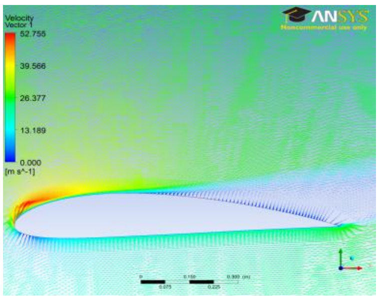
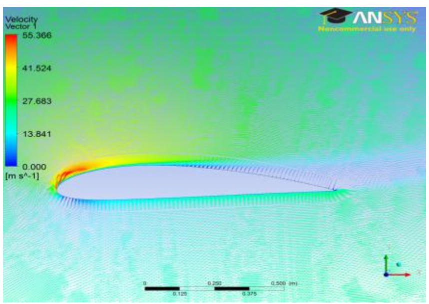
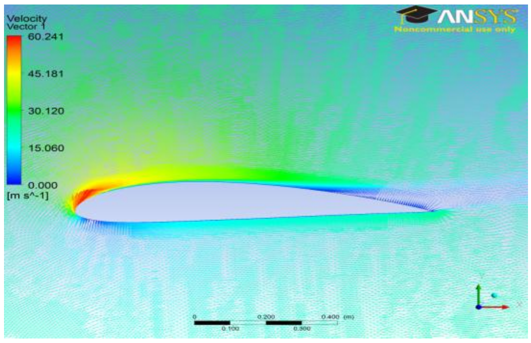


Figure 6 Flow separation at 17*o* angle of attack for a clean airfoil



(a) Trailing edge distributed suction



(b) Leading edge distributed suction

Figure 7 Flow separation at 17*o* angle of attack for trailing and leading edge distributed suction

IV. DIMPLES

Armin Ghoddoussi et al [2011] has done the work on the inward dimples analysis matrix is presented, dimples are located after and before the maximum thickness to preserved laminar flow closer to leading edge. Observation show that as the dimple move towards leading edges maximum lift is going to decrease and drag is going to increase and also there is no noticeable change compare to plain airfoil and in stall condition.

Deepansu srivastav et al [2012] presented work on the inward and outward dimples on NACA0018.

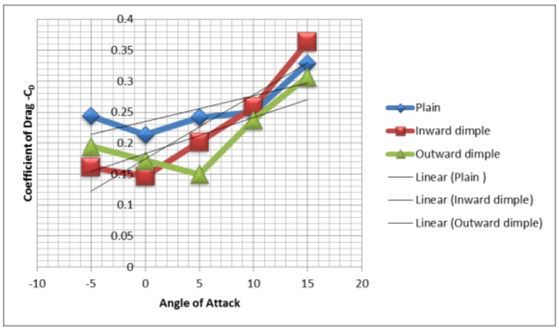


Figure 8: Plot of Coefficient of Drag versus Angle of attack for different configurations.

From the fig. it is clearly indicated that the outwards dimples has noticeable decrement in drag coefficient. In the outwards dimples experiment is done on two different shape 1. Round shape (RD) 2. Composite dimples (CtD).

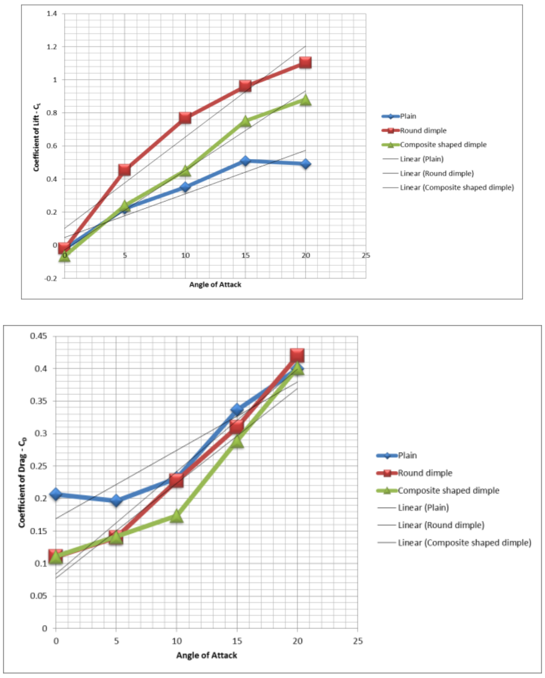


Figure 9 (a) & (b): Plot of Coefficient of Lift and Drag versus Angle of attack for different configurations.

Fig. 9(a) shows that RD can increase value of lift coefficient compare to CtD at different angle of attack, where fig. 9(b) shows that CtD can decrease value of drag coefficient compare to RD, but overall aerodynamic efficiency of RD is more than CtD.

V. PULSE JET

Shutian Deng et al [2007] done work on the pulse jet, the set up with three cases 1. Baseline case 2. Pulsed blowing 3. Blowing with 30° pitch and 90° skew angle

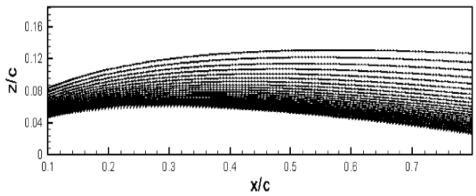


Fig.10. Mean velocity vectors.

Case 1 is describe in [10]. In case two blowing velocity has 90° pitch angle and 0° skew angle, comparison of mean velocity vector of fig () with fig() show that reversed flow is completely eliminated and flow is completely reattached to the surface, though this pressure coefficient distribution will not improve the lift. It reduce both lift as well as drag.

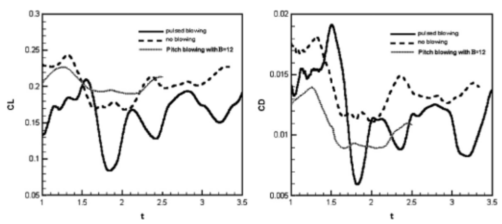


Fig. 11. Temporal variations of lift and drag coefficients

So in third case they do experiment with 30° pitch and 90° skew angle and result show that it reduce the drag while lift is maintain as in baseline case. By selecting proper blowing mass, frequency, pitch and skew angle result can be improved.

Kianoosh Yousefi et al [2013] done experiment on Numerical Study of Flow Separation Control by Tangential and Perpendicular Blowing on the NACA 0012 Airfoil, result show that in tangential blowing by changing blowing amplitude and coefficient lift to drag ration can be increase. Best result is achieve at 0.5 blowing amplitude and 0.0875 blowing coefficient. In perpendicular blowing lower blowing amplitude and coefficient give somewhat good result than baseline case. It can change stall from 14 to 16 while no significant of tangential blowing on stall.

In the slotted airfoil high lift is achieved by putting slot in airfoil so that high energy air can pass through slot from lower surface to upper surface and able to energized upper surface boundary layer so that separation can be reduce.[5]

Gottlieb et al [1995] done the experiment on the NACA 6 series airfoil with leading edge slot, result show that leading edge slot increase 0.6 in maximum lift and stall can delay by 14°.

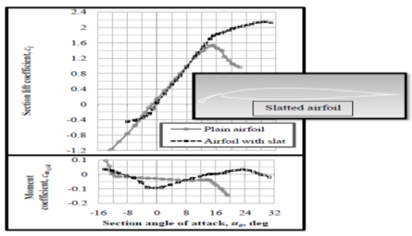


Figure 12. Section lift coefficient and pitching moment coefficient against angle-of-attack of NACA 641-212

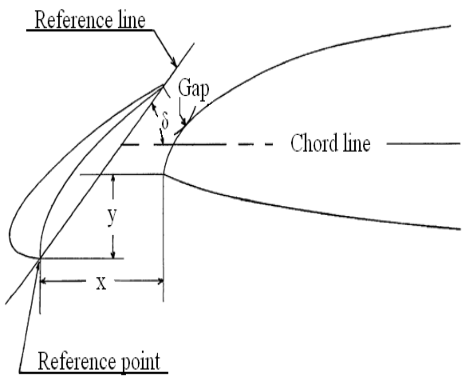


Figure 13. NACA 641-212 with the leading-edge slat geometry notation

Summary Of Maximum Lift Characteristics And Leading-Edge Slat Configuration For Naca 641-212 Airfoil At The Reynolds Number 6.0 X 106

	$C_{l\,m}$	$\propto_{cl\,m}$	x(% chord)	Y(% chord)	δ $(de$	Gap(% chord)
Plain airfoil	1.55	15	-	-	-	-
Slotted airfoil	2.15	29	9.9	-6.3	43.3	1.7

Joon W. Lim et al has done work on the application of slotted airfoil in helicopter results show that slot can increase thrust by 25%, but drag penalty observe al low angle of attack. They also compare wide chord airfoil with slotted airfoil, for low thrust level wide chord airfoil is beneficial then slotted. While for high thrust level slotted airfoil is beneficial then wide chord.

C.N. Nayeri et al[2010] done comparison between baseline case and slat configuration, And found that lift coefficient is going to increase and drag coefficient going to decrease

VI. VORTEX GENERATOR

First time conventional passive vortex generator was developed by Taylor in 1947 to prevent flow separation. First systematic study done by schubauer and spangenberg in 1950. The vortices created by vortex generator transfer lower energy air to main stream and high energy air to the surface so it can resist adverse pressure gradient effect, can able to prevent flow separation.

Li jiang et al [2007] done numerical simulation on NACA0012 with passive vortex generator and active vortex generator. In passive vortex generator two types and fully deployed to their maximum height quarter circle type vortex generator are located. From fig.[] indicate that compare to baseline case flow separation can be reduce but in between 0.05 to 0.1 x/c and again separated in between 0.2 to 0.25 x/c , while in active vortex generator separation is completely eliminated.

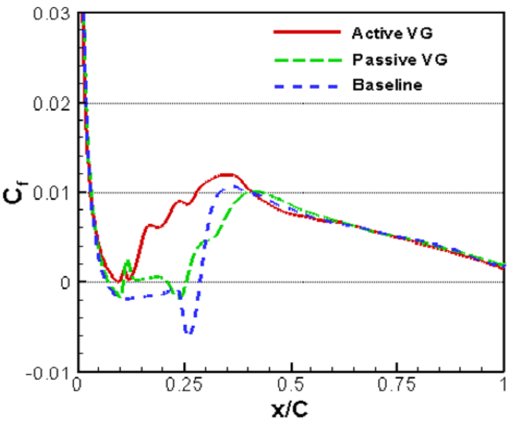


Fig. 14. Mean skin friction coefficient of Case 2 in comparison with baseline and Case 1.

Nowak and solies has done new concept of delta flap vortex generator to increase more lift and to delay stall, it located at leading edge and size of 0.3c. it's optimum position can increase lift in some range of angle of attack

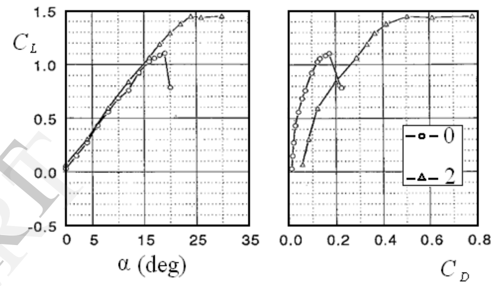


Figure 15. Typical result of a wind-tunnel test

VII. DIFFERENT CFD MODELS

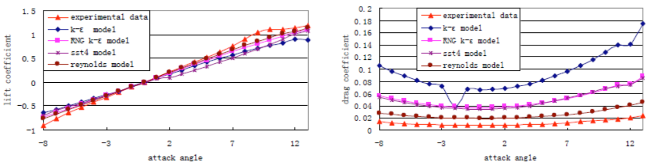
For the numerical simulation different models are available in software package. Depending upon required result parameter, flow condition ant it parameter value these model will be selected for analysis.

If we select inviscied model then shear stress part will not consider, so for the flow separation analysis at different angle of attack, it will not applicable. For that laminar and turbulent models are there. In fluent if laminar modal is selected and Reynolds number is in the transition rang, that time fluent will not give u error regarding turbulent model but if CFX is there that time error regarding turbulent is display.

Turbulent model are presented basies on velocity fluctuation, flow mass, momentum, and energy transferred by fluctuating velocity and mixing there quantities. Regarding this turbulent model and it's description and application described in [15].

Ji vao et al [2012] work on influence of different turbulent model on vertical axis wind turbine, result show that influence of different velocity is less but on pressure is noticeable. k-ε model is not applicable for the adverse pressure gradient effect for that k-ω, SST and other modal are applicable. In SST out of presented sub mode gamma-theta modal is best presented in [14].

Minjun et al [2012] present work on performance of NACA0018 wind turbine airfoil using different five turbulent model and found that in lift coefficient result among that five there is minor variation but in drag coefficient much more variation. Out of five Reynolds model is good.



(a) lift coefficient (b) drag coefficient

Fig 4 variation of the lift coefficient and drag coefficient with the change of the attack angle

VIII. CONCLUSION

The summary of the present literature review is as follows:

1. Disturbance propagate in the form of acoustic wave

2. Most of the report reduce the flow separation using different technique in improve lift and drag coefficient. to reduce flow separation boundary layer is energized using either of above technique

3. To improve the performance of airfoil either lift coefficient must be increase or drag should be decrease

4. By reducing flow separation or preventing it, there is no surety that lift coefficient is increase, there is reduction in drag and by reducing flow separation stall is going to delay

5. To improve lift coefficient pressure coefficient must me properly distributed on airfoil surface

6. Ever CFD model has owned specification for the specific case, CFD modals selected as per that

REFERENCES

[1] John D Anderson Jr, *Introduction to flight* . McGraw-Hill Book Company, sixth ed.2012.
[2] E.L.Houghton, P.W.Carpenter. Aerodynamic for Engg. Student. CBS fourth ed. 2005
[3] Yunus Cengel, *Fluid mechanic* . McGraw-Hill Book Company, sixth ed.,2009
[4] F. M. White, *Fluid Mechanics*. McGraw-Hill, fifth ed., 2003.
[5] E. Hau. Wind Turbines: Fundamentals, Technologies, Applications, Economics (2nd Edition). Springer, 2006
[6] Rong Ma, Peiqing Liu. "Numerical Simulation of Low-Reynolds-Number and High-Lift Airfoil S1223". Proceedings of the World Congress on Engineering 2009 Vol II
[7] S.Kandwal , Dr. S. Singh. "Computational Fluid Dynamics Study Of Fluid Flow And Aerodynamic Forces On An Airfoil" International Journal of Engineering Research & Technology (IJERT), Vol. 1 Issue 7, September - 2012
[8] Ji Yao, Jianliang Wang,Weibin Yuanb, Huimin Wang, Liang Cao. "Analysis on the influence of Turbulence model changes to aerodynamic performance of vertical axis wind turbine". Procedia Engineering 31 (2012) 274 – 281
[9] Ji Yaoa, Weibin Yuan, jianliang Wang, Jianbin Xie , Haipeng Zhou , Mingjun Pengd, Yong Sun. "Numerical simulation of aerodynamic performance for two dimensional wind turbine airfoils". Procedia Engineering 31 (2012) 80 – 86
[10] Evan Kontras, Kyle Gould, Davide Maffeo. " The NACA 4212 airfoil was analyzed using three separate methods". MAE 4440/7440 Aerodynamics University of Missouri
[11] Hua Shan, Li Jiang, Chaoqun Liu "Direct numerical simulation of flow separation around a NACA 0012 airfoil" Computers & Fluids 34 (2005) 1096–1114
[12] Sanjay Mittal*, Priyank Saxena "Hysteresis in flow past a NACA 0012 airfoil" Comput. Methods Appl. Mech. Engrg. 191 (2002) 2179–2189
[13] M. Goodarzi, R. Fereidouni and M. Rahimi "Investigation of flow control over a NACA 0012 airfoil by suction effect on aerodynamic characteristics" Canadian Journal on Mechanical Sciences & Engineering Vol. 3 No. 3, June 2012
[14] Apostolos Tentolouris Piperas, Martin O.L. Hansen. "Investigation of Boundary Layer Suction on a Wind Turbine Airfoil using CFD". Wind Energy Building 403 Kongens Lyngby *Master's Thesis* 5th August 2010
[15] Armin Ghoddoussi. "A CONCEPTUAL STUDY OF AIRFOIL PERFORMANCE ENHANCEMENTS USING CFD". Submitted to Department of Aerospace Engineering and the faculty of the Graduate School of Wichita State University in partial fulfillment of the requirements for the degree of Master of Science(2011)
[16] Deepanshu Srivastav "Flow Control over Airfoils using Different Shaped Dimples" IPCSIT vol.33(2012)©(2012) IACSIT Press, Singapore
[17] Shutian Deng, Li Jiang, Chaoqun Liu. "DNS for flow separation control around an airfoil by pulsed jets". Computers & Fluids 36 (2007) 1040–1060
[18] Kianoosh Yousefi, S.Reza Saleh & Peyman Zahedi. "Numerical Study of Flow Separation Control by Tangential and Perpendicular Blowing on the NACA 0012 Airfoil" International Journal of Engineering (IJE), Volume (7) : Issue (1) : 2013
[19] Abbott, I. H., and von Doenhoff., A. E., *Theory of Wing Sections*, Dover, New York, 1959
[20] S. M. Gottlieb, "Two-Dimensional Wind-Tunnel Investigation of Two NACA 6-Series Airfoils with Leading-Edge Slats," *NACA RM No. L8K22*, Langley Aeronautical Laboratory Langley Field, VA, Jan 1949.
[21] JoonW. Lim. "Application of a Slotted Airfoil for UH-60A Helicopter Performance" NASA Ames Research Center Moffett Field, California (2002)
[22] Hua Sha, Li Jiang, Chaoqun Liu, Michael Love, Brant Maines. "Numerical study of passive and active flow separation control over a NACA0012 airfoil" . Computers & Fluids 37 (2008) 975–992
[23] D. K. Nowak and U. P. Solies, "Wind-Tunnel Tests of a High Lift Generation and Stall/Spin Recovery System," *Journal of Aircraft*, vol. 37, pp. 383-389, 2000