

## Flow Control over Airfoils using Different Shaped Dimples

Deepanshu Srivastav

BE (Hons.)

Email: [deepanshu@gmail.com](mailto:deepanshu@gmail.com)

Phone number: +91-9049557547

BITS Pilani, K K Birla Goa campus  
Zuarinagar Goa, India - 403726

**Abstract.** The present work describes change in aerodynamic characteristics of an airfoil by applying certain surface modifications in form of dimples. At first surface modifications that are considered here are outward and Inward dimples on the wing model. A 2-Dimensional computational fluid dynamics CFD is done on both using k-w turbulence model, thereafter based on its results the better of the two is chosen. After choosing the better dimpled configuration, different shaped dimples are tested and compared to the plane airfoil model. This CFD analysis is done in 3-D by taking a segment of the airfoil with one dimple on it. A comparative study showing variance in lift and drag of modified airfoil models at different angle of attacks is done. The results were in agreement that dimples on the surface aircraft wing model doesn't affect the pressure drag much since it is already aerodynamic in shape but it can affect its aerodynamics when the airfoil is at different angle of attacks. This project verifies if the dimples that reduce a golf ball's drag, can also alter flow dynamics around airfoil for better aerodynamic efficiency. Dimples delay the boundary layer separation by creating more turbulence over the surface thus reducing the wake formation. This in turn reduces drag drastically. The airfoil profile considered in the present study is NACA-0018 with uniform cross-section throughout the length of airfoil. Subsonic flows are considered for the study. The CAD model is prepared in CatiaV5 R18 and simulations are carried out in Comsol 3.4 and Comsol 4.2a. The overall aim of the study is improved maneuverability and performance of an aircraft by flow manipulation over the airfoil. The results justify the increase in the overall lift and reduction in drag of the airfoil.

**Keywords:** Airfoil, Angle of attack, Aerodynamic efficiency, Drag and Lift.

### 1. Introduction

From the beginning of human race, man has always dreamt of flying and on December 17, 1903 Wright brothers gave human race new wings and hoped for continuous endeavours in this field. Now we have progressed to great extent in air but still after so much has been done there are certain constraints binding us. Freedom in the air is still not complete. Continuous attempts are being made to increase freedom in air, be it speed, size or manoeuvrability. From commercial jetliners to supersonic fighters, there has been an exponential growth in the aviation industry. Still there is vast scope for further improvements. Here is a study that makes one such attempt.

At present, different kinds of surface modifications are being studied to improve the manoeuvrability of the aircraft. Vortex generators are the most frequently used modifications to an aircraft surface. Vortex generators create turbulence by creating vortices which delays the boundary layer separation resulting in decrease of pressure drag and also increase in the angle of stall. It helps to reduce the pressure drag at high angle of attack and also increases the overall lift of the aircraft. Riblets are another type of modification that is being considered these days.

The surface modifications which are being considered in the given study are dimples of types and shapes. Till now these have been ignored because dimples help in reduction of pressure drag. In case of aerodynamic bodies pressure drag is very little compared to bluff bodies. An airfoil is an aerodynamic body so dimples do not affect to its drag much at zero angle of attack, but as soon as airfoil attains some angle of attack, wake formation starts due to boundary layer separation. Application dimples on aircraft wing model works in same manner as vortex generators. They create

turbulence which delays the boundary layer separation and reduces the wake and thereby reducing the pressure drag. This also assists in Lift of the aircraft. Most importantly this can be quite effective at different the angle of attacks and also can change angle of stall to a great extent. A stall is a condition in aerodynamics and aviation where the angle of attack increases beyond a certain point such that the lift begins to decrease. The angle at which it occurs is called the critical angle of attack or angle of stall. Flow separation begins to occur at small angles of attack while attached flow over the wing is still dominant. As angle of attack increases, the separated regions on the top of the wing increase in size and hinder the wing's ability to create lift. At the critical angle of attack, separated flow is so dominant that further increases in angle of attack produce less lift and vastly more drag.

In order to verify the effect of dimples, the following computational study has been made starting from 2-D study of inward and outward dimpled airfoil. Based on the drag analysis, outward dimples were chosen due to better performance. Thereafter 3-D study of NACA0018 with and without dimples with different shapes is done. In order to do simulation in 3-D, a slice of the airfoil was simulated with and without single dimple for a comparative study. Different kinds of dimples were considered. Afterwards multiple dimples were considered for the simulation but these results are not included in the paper.

The long-term goal is to design and manufacture optimal vortex generators, ‘outward dimples’ that can produce vortices of prescribed strength and duration for the real-time control of aerodynamic flows that are either undergoing transition or are fully turbulent, attached or separating. Through this study we aim at making aircrafts more manoeuvrable by dimpled airfoils. Also we are looking to improving performance by more L/D ratio i.e. increasing aerodynamic efficiency. Aerodynamic efficiency is one of the key parameters that determines the weight and cost of an aircraft. Roughly speaking, an aircraft's range is directly proportional to its aerodynamic efficiency without any increase in fuel usage. Improved aerodynamics is critical to both commercial and military aircraft. For commercial aircraft, improved aerodynamics reduces operating costs. It also significantly contributes to the national security by improving efficiency and performance of military aircraft.

## 2. Description

### 2.1. Model

Contributions In order to verify the proposed idea, an airfoil profile needed to be selected on which whole study will be based. This is a conceptual study which assumes an Incompressible and isothermal flow. All the simulations are carried out on NACA 0018 (chord length-16 cm)[2] .

NACA0018 is a symmetrical airfoil as shown in Fig. 1. symmetrical airfoils produce less lift than cambered airfoils but assists more in aerobatics and maneuverability of an aircraft.



Figure 1: Two dimensional NACA 0018 profile made by using X and Y coordinates.

Steady State analysis is considered here assuming Turbulent flow. Reynolds number suggests the flow to be fully turbulent.

$$Re = \frac{\rho \cdot u \cdot l}{\mu} \approx 320000$$

CAD model development is done on Catia V5R18 and Solidworks 2009. Simulations are carried out in Comsol 3.5 and Comsol 4.2a.

#### Turbulence Models:

**k-ε turbulence model** is used for 3-D as well as 2-D simulations of airfoil segment in COMSOL fluid flow module to predict the flow regime. The k-ε model is one of the most common turbulence models. 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 factor,  $\epsilon$ . The latter is a variable that determines the scale of the turbulence, whereas the first variable,  $k$ , determines the energy in the turbulence.

**k- $\omega$  turbulence model** is used for 2-D simulation of airfoil segment in COMSOL fluid flow module to predict the flow regime. The k- $\omega$  model is also two equation model. The first transported variable is turbulent kinetic energy,  $k$ . The second transported variable in this case is the specific dissipation,  $\omega$ . It is the variable that determines the scale of the turbulence, whereas the first variable,  $k$ , determines the energy in the turbulence. k- $\omega$  model used for adverse pressure gradient problems.

Following equations govern the flow profile around the models simulated:

**Continuity equation:**

$$\nabla \cdot \mathbf{u} = 0$$

**Navier Stokes:**

$$\rho \mathbf{u} \cdot \nabla \mathbf{u} = \nabla \cdot [-p\mathbf{I} + (\eta + \eta_T)(\nabla \mathbf{u} + (\nabla \mathbf{u})^T)] + \mathbf{F}$$

**Transport equations for k- $\epsilon$  model:**

$$\rho \mathbf{u} \cdot \nabla k = \nabla \cdot [(\eta + \eta_T/\sigma_k) \nabla k] + \eta_T P(\mathbf{u}) - \rho \epsilon$$

$$\rho \mathbf{u} \cdot \nabla \epsilon = \nabla \cdot [(\eta + \eta_T/\sigma_\epsilon) \nabla \epsilon] + C_{\epsilon 1} \epsilon \eta_T P(\mathbf{u})/k - C_{\epsilon 2} \rho \epsilon^2/k$$

Where:  $P(\mathbf{u}) = \mathbf{u} \cdot (\nabla \mathbf{u} + (\nabla \mathbf{u})^T)$ , and  $\eta_T = \rho C_\mu k^2/\epsilon$   
 $C_{\epsilon 1}=1.44, C_{\epsilon 2}=1.92, C_\mu=0.09, \sigma_k=1.0, \sigma_\epsilon=1.3$

**Transport equations for k- $\omega$  model:**

$$\rho \mathbf{u} \cdot \nabla k = \nabla \cdot [(\eta + \eta_T \sigma_k) \nabla k] + \eta_T P(\mathbf{u}) - \beta_k \rho k \omega$$

$$\rho \mathbf{u} \cdot \nabla \omega = \nabla \cdot [(\eta + \eta_T \sigma_\omega) \nabla \omega] + a \omega \eta_T P(\mathbf{u})/k - \beta \rho \omega^2$$

Where:  $P(\mathbf{u}) = \mathbf{u} \cdot (\nabla \mathbf{u} + (\nabla \mathbf{u})^T)$ , and  $\eta_T = \rho k/\omega$   
 $\beta_k=0.09, \beta=0.075, a=0.55$

**Solver used:** Segregated solver – It breaks the non-linear equations into indefinite small linear equations. Generalized minimal residual method (usually abbreviated GMRES) method is used to solve each of these linear equations. GMRES is an iterative method for the numerical solution of a system of linear equation. Also PARDISO package is used, which is a thread-safe, high-performance, robust, memory efficient and easy to use software for solving large sparse symmetric and unsymmetric linear systems of equations on shared-memory and distributed-memory multiprocessors. Maximum number of Iterations are set to 100 but changed if model doesn't converge until 100<sup>th</sup> iteration. Tolerance value is given 1.0e-03. Solution converges when error value reaches 1.0e-05.

**Meshing:** Triangular meshing is used for the analysis. Each model is first simulated at coarse meshing. After every converged solution the meshing is refined and again the model is simulated. This continues until maximum memory usage limit is reached for the computer system. In this case solution doesn't converge and previously meshed results are considered for that specific model.

All simulations of NACA 0018 are carried out at different angles of attack, taking inlet velocity ( $u_x$ ) as 20m/s i.e. 72km/hr.  $u_y$  and  $u_z$  are taken to be zero. One of the objectives of this computational study is also to shorter the takeoff distance of the aircraft by creating sufficient lift with minimum drag at low velocity. For this reason airfoil model is simulated at such low velocity.

In order to analyze the complete velocity and pressure profiles different domains are considered according to different sizes and orientations of the airfoil. Density of the air is taken as 1.2 kg/m<sup>3</sup> and Dynamic viscosity as 1.2e-5 Pa-s. Domains are all cuboidal shaped with all side walls of the airfoil given symmetrical boundary condition except for the inlet and outlet walls. Chord length of the airfoil is 16 cm and the span is kept constant at 0.8 cm in all simulations.

Inlet conditions for turbulence are Turbulent length Scale ( $L_T$ ) = 0.01m, Turbulent Intensity ( $I_T$ ) = 0.05 or Turbulent Kinetic energy ( $k_0$ ) = 0.005m<sup>2</sup>/s<sup>2</sup> Specific Turbulent dissipation rate ( $w_0$ ) = 10s<sup>-2</sup>, Turbulent dissipation rate ( $k_0$ ) = 0.005m<sup>2</sup>/s<sup>3</sup>.

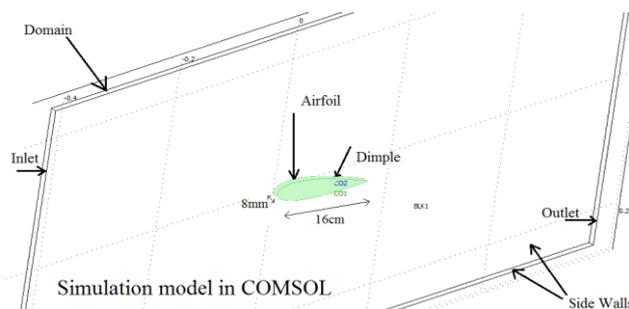


Figure 2: Complete 3D model with shown domain and boundaries to simulate airfoil in COMSOL.

First 2-D simulation is carried out to draw comparison between Inward and Outward dimples, shown in Fig. 3. Also both are compared to plain airfoil NACA0018 without any dimple. Analysis is done at -5,0,5,10 and 15 degrees of angle of attack. Drag analysis is considered for comparison.

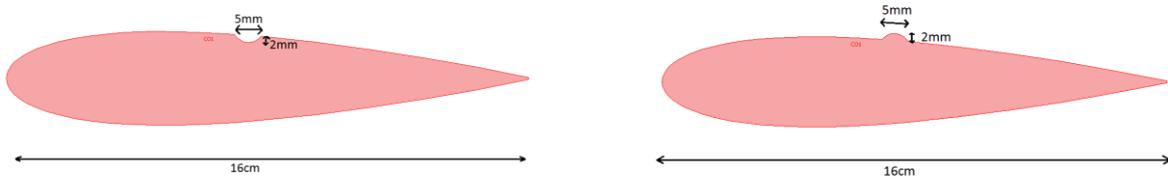


Figure 3: NACA 0018 with Inward and Outward dimple.

After this 3-D plain airfoil model is simulated for the given conditions to be used as a reference. Two other models are simulated with two different outward shaped dimples. The Round shaped dimple (RD) (Fig. 4.) having semi hemispherical shape and Composite dimple (CtD)(Fig. 5.) having leaf like dome shape are considered for the study. Also a configuration with vortex generator (Fig. 4.) is simulated to draw comparison between presently used configurations with the suggested configurations.

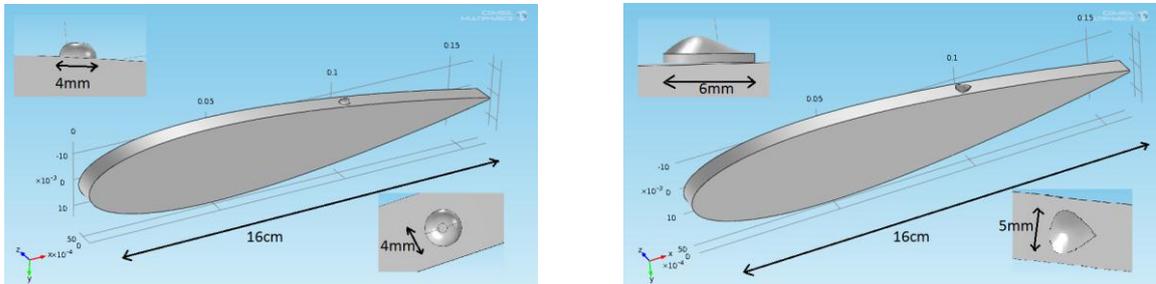


Figure 4: NACA 0018 with Round and Composite dimple.

All airfoil models are tested at 0, 5, 10, 15 and 20 degree of angles of attack. Vertical and horizontal forces i.e. Lift and Drag are calculated using Comsol post processing. Both lift and drag represented through dimensionless numbers Coefficient of Lift ( $C_L$ ) and Coefficient of Drag ( $C_D$ ).

The performance of each model is measured based on the aerodynamic efficiency i.e. Lift to drag ratio ( $L/D$ ). Aerodynamic efficiency is one of the key parameters that determines the weight and cost of an aircraft. Roughly speaking, an aircraft's range is directly proportional to its aerodynamic efficiency without any increase in fuel usage. Also the effect on the change of stalling angle of these configurations compared to plain airfoil is also analyzed closely.

## 2.2. Results and Discussion

The study starts with CFD analysis of 2-D NACA0018 airfoils with inward and outward dimples. Coefficient of drag is compared of both of these configurations along with one of plain airfoil. The study shows (Fig. 5.) that outward dimple produces lesser drag at positive angle of attacks and therefore more suitable for further studies.

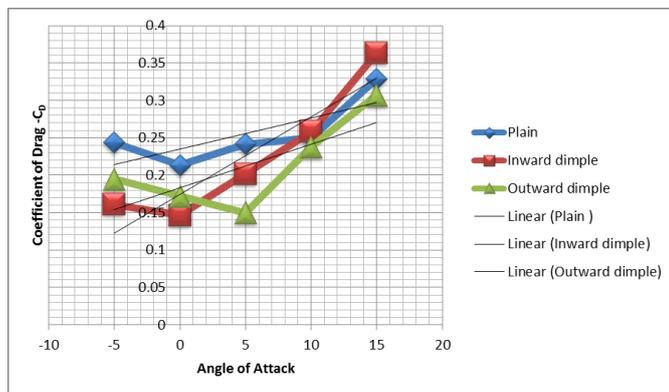


Figure 5: Plot of Coefficient of Drag versus Angle of attack for different configurations.

Now in 3-D study is done of outward dimples both the models RD and CtD configurations seems to create turbulence to delay boundary layer separation. Turbulence as we know is random movement of particles in a medium. When this random motion is in form of circular motion, we call it as Vortex. This phenomenon of vortex formation is called Vorticity. The effect of this random motion can clearly be seen through the change in Lift and Drag of the systems.

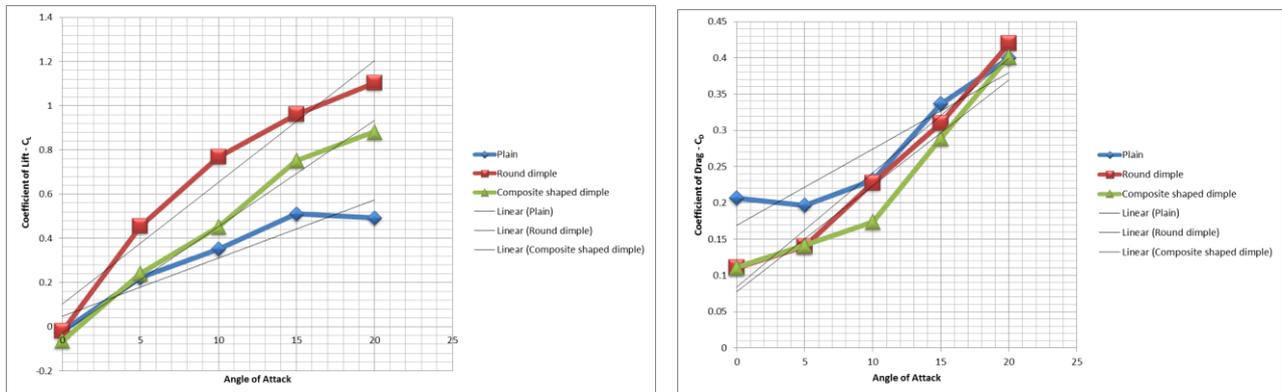


Figure 6: Plot of Coefficient of Lift and Drag versus Angle of attack for different configurations.

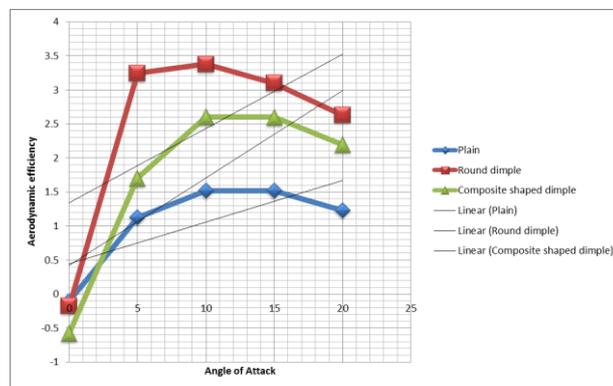


Figure 7: Plot of L/D ratio versus Angle of attack for different configurations

Coming back to complete airfoil study Fig.6. and Fig.7. show changes in lift and decrease in drag as the angle of attack increases.

Figure. 6. shows effect of angle of attack on Coefficient of Lifts of all configurations. Both RD and CtD show more lift than a Plain airfoil configuration at corresponding angles of attack. Also RD and CtD models shows uniform increase in lift throughout all angles of attack considered for the study. Figure. 6. Also shows variation in coefficient of Drag with respect to angle of attack on the figure on right hand side. Both dimpled model (RD and CtD) show decrease in drag than plain airfoil model. Out of RD and CtD, CtD shows least drag.

Talking of aerodynamic efficiency as a whole, which shows combined effect of lift and drag is shown through Fig 7. through L/D ratio. RD model shows highest L/D ratio i.e. aerodynamic efficiency. All modified models show higher aerodynamic efficiency than Plain airfoil at all angles of attack. All four models show an increase in the L/D ratio till they attain a maximum value and thereafter it decreases.

Overall analysis favors the use of dimples. Based on this result a new model multi dimpled (MD) (Fig.8.) is suggested. This model will use a matrix [m,n] of dimple like one shown in Fig.8. [1,10]. This matrix of dimples will smart i.e. it will arrange itself over the airfoil by sensing the boundary layer separation at different points. This sensing of boundary layer separation will be done through MEMS based pressure sensors which will estimate the pressure gradient. Through a predefined algorithm using this data for MEMS pressure sensors boundary layer separation can be predicted. The same algorithm will then send a command to rearrange the dimple matrix to optimize for maximum aerodynamic efficiency. The complete concept and explanation of dimple matrix is not mentioned in the present paper.

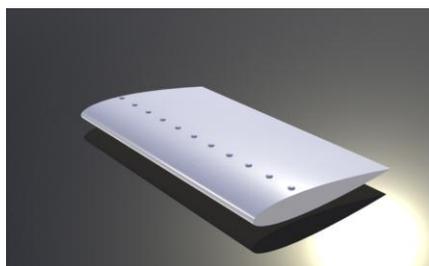


Figure 8: Multi dimpled configuration

### 2.3. Conclusions

Addition of dimples has proven to be effective in altering various aspects of the flow structure. With such significant flow structure the resultant lift and drag forces are also altered. Primarily the outward dimples are proved to be most suitable as proved by the results of this study. Based on these results a new Smart dimple matrix is suggested over airfoils which will sense boundary layer separation and arrange dimple in the least drag and high lift configuration.

Downright, the total aerodynamic efficiency of the airfoil is improved by this complete idea. But for complete verification of its effect an experimental study is required. Also full scale testing focusing on aerodynamic force measurement is essential to determine the sustainability of the outward dimples to act as highly efficient vortex generators.

The concept is very new and with the implementation of dimple matrix, it could be extremely beneficial in making an aircraft more manoeuvrable by changing flow characteristics. Also it increases the aerodynamic efficiency and therefore helps in improving the performance also. The idea will also assist in shorter take-offs at low speed.

### 3. Acknowledgements

My sincere acknowledgments to BITS Pilani K K Birla Goa Campus, Dr. K N Ponnani, Prof. Srinivas Krishnaswamy and my friends for their support and guidance.

### 4. References

- [1] MICHAEL B. PATACSIL., “Dimples on Wings”, California Science Fair, 2010
- [2] University of Illinois at Urbana-Champaign (UIUC) Airfoil database, [http://www.ae.illinois.edu/m-selig/ads/coord\\_database.html](http://www.ae.illinois.edu/m-selig/ads/coord_database.html)
- [3] CHANA R. and VISHWANATHI, P. R., “Base Drag Reduction Caused by Riblets on a GAW(2) Airfoil”, National Aerospace Laboratories, Journal of Aircraft, Volume 35. No. 6, November-December 1998
- [4] RENEAUX, J., “Overview on drag reduction Technologies for Civil transport aircraft”, European Congress on Computational Methods in Applied Sciences and Engineering ECCOMAS, 2004.
- [5] ZERIHAN, J., “An Investigation into the Aerodynamics of Wings in Ground Effect”, PhD Thesis, university Southampton, School of Engineering, 2001.
- [6] MAHON, S. and ZHANG, X. “Computational analysis of pressure and wake characteristics on an aerofoil in ground effect”, ASME, 2005, 127, PP 290-298
- [7] RAZENBACH, R. and BARLOW, J.B., “Two dimensional aerofoil in ground effect, an experimental and computational study”, SAE, 1994, SAE 942509.
- [8] DAVIES, J.M., “The aerodynamics of golf balls”, J of Applied Physics, 1949, 20, (9), PP 821-828.
- [9] STORMS, B.L., “Lift enhancement of an aerofoil using a gurney flaps and vortex generators”, J Aircr, 1994, 31, (3), PP 542-547.
- [10] NICKERSON, J.D., “A study of vortex generators at low Reynolds numbers”, AIAA, 1986, AIAA-86-0155.
- [11] BEVES, C.C. BARBER, T.J. and LEDNARDI, E., “Aerofoil flow separation suppression using dimples”, The Aeronautical Journal, VOL. 115 No 1168, Pg 335-344, June 2011.
- [12] GUGLIELMO, J.J. and SELIG M.S., “Spanwise variations in profile drag for airfoils at low Reynolds numbers”, J Aircr, 1996, 33, (4).
- [13] MAMOU, M., YUAN, W., KHALID, M., WOKOECK, R. and RADESPIEL, R., “Transition prediction in low Reynolds airfoil flows using finite element method coupled with the eN method”, 25th AIAA Applied Aerodynamics Conference, San Francisco, 5-8 June, 2006.
- [14] SCHLICHTING, H. Boundary-Layer Theory, 6th edition, McGraw-Hill, New York, 1968.
- [15] LAMBERT, S., MCKEON, B.J., KOBERG, W.H. and MORRISON, J.F. Fundamental Studies of Active Dimples, Turbulent Shear Flow Phenomena-4, Williamsburg, 2005, pp 283 – 288.
- [16] LAMBERT, S. and MORRISON, J.F., Fundamental Studies of Active Dimples, 2006, AIAA-2006-3182.
- [17] WU, J.Z. and WU, J.M., Vorticity dynamics on boundaries, Adv. in Appl Mech, 1996, 32, pp 119-275.