Role of single bump over the surface of E398 Airfoil to improve the aerodynamic performance

Asif Ali Laghari
asifali@quest.edu.pk
Quaid-e-Awam University college of Engineering, Science & Technology, Larkana, Pakistan

Shafiq-ur-Rehman
bshafiqpn@hotmail.com
Pakistan Navy Engineering College, Karachi, Pakistan

M. Tarique Bhatti
trqbhatti@hotmail.com
Quaid-e-Awam University college of Engineering, Science & Technology, Larkana, Pakistan

Allah Rakhio
aftabgolo@yahoo.com
State Key Laboratory of Mechanical Transmission Chongqing University, China

Manthar Ali Khoso
mantharkhoso@gmail.com
Quaid-e-Awam University college of Engineering, Science & Technology, Larkana, Pakistan

Sajjad Bhangwar
sajjadbhangwar@gmail.com
Quaid-e-Awam University college of Engineering, Science & Technology, Larkana, Pakistan

ABSTRACT

Flow Separation over the surface of the airfoil affect the aerodynamic performance of the airfoil, flow separation increases drag and decreases lift. Aerodynamic efficiency of an airfoil can be improved by delaying separation, which increases the lift force and reduces the drag force. Flow separation can be controlled by altering boundary layer behavior such as reducing boundary layer thickness and increases maximum velocity over the surface of an airfoil, these can be done by modifying the geometry of airfoil or other active flow control techniques and passive flow control techniques. In this research, the E398 airfoil was selected for analysis. Analysis has been carried out on smooth and modified E398 airfoil, airfoil modified with single bump centered at 0.05C, 0.1C, 0.15C, 0.2C, and 0.3C from leading edge, a chord length of the airfoil is 0.128m. A numerical study has been carried out by using Ansys Fluent 14.0 over Smooth and Uni-Bump E398 airfoil, at low Reynolds Number ranging from 18000 to 54000 and Angle of Attack from 0^0 to 15^0. Numerical Results show that the aerodynamic efficiency of a modified airfoil with bump centered at 0.1C is higher as compared to smooth airfoil at Reynolds 18000 and 36000. When Reynolds number is 54000, aerodynamic efficiency of the smooth airfoil is higher than the modified airfoil with a single bump at 0.1C. Analysis has been performed by changing the location of a bump from 0.5C to 0.3C aerodynamic performance decreases as bump shifted toward the maximum thickness of airfoil because suction pressure decreases on the upper surface of an airfoil. It is observed that as a bump near leading edge of the airfoil, shows the higher aerodynamic performance, the suitable locations of the bump are at 0.5C and 0.1C from leading edge.

Keywords— Flow separation, E398 Airfoil, Low reynolds number, Aerodynamic performance

1. INTRODUCTION

Chances of flow separation at low Reynolds number are higher as compared to high Reynolds number. A lot of research work has been carried out at low Reynolds number in the recent past to improve the performance of unmanned aerial vehicles (UAVs) and micro Aerial vehicles (mAVs). These aircraft are used for military surveillance and communication and planetary exploration i.e. Mars [1]. These aircraft operate at normal and high altitudes during flight operation, the performance of an aircraft depends on the growth of the boundary layer on solid surfaces and its interaction with the fluid flow [2].

When a real fluid flows over the solid surface, a layer is formed in which viscous forces are dominant is known as the boundary layer. From a solid surface to the point where the velocity of fluid reaches to the $0.99U\infty$ is called boundary layer thickness (δ). Boundary layer thickness varies from leading edge to trailing edge. When the pressure gradient is positive $\left(\frac{\partial p}{\partial x} > 0\right)$ boundary layer thickness increases from leading edge to trailing edge. Due to adverse pressure gradient velocity of fluid near solid surface approaches to zero after that point flow departed from solid surface, region is known as separated flow region [3]. When its magnitude of adverse pressure is to small then boundary layer may reattach and transition occurs after the laminar separation because of it a turbulent boundary layer formed over the surface of airfoil and it resist the separation. Flow separation increases the form

drag force and decreases the lift force which adversely affects the aerodynamic efficiency of airfoil. The aerodynamic efficiency of aircraft can be improved by (a) altering boundary layer behavior over surface of the airfoil (the b) modify the geometry of airfoil [1].

Continuous efforts have been made to increase the aerodynamic efficiency of the airfoil in this regards different techniques studied to investigate the performance of different airfoil. Sometimes boundary layer behavior changes by co-flow jet, Dielectric barrier discharge, vertex generator etc. are known as Active flow control techniques [4,5]. When fluid flows over an airfoil at low Reynolds number, efficiency can be improved by placing large-scale roughness on the surface of airfoil this technique is known as passive flow separation control technique [3]. Different research has been published earlier, to investigate the effect of the boundary layer with the addition of large-scale roughness such as single and multi-bumps at a different location either near leading and trailing edge. Roughness near leading edge shows better performance to delay flow separation and increases aerodynamics efficiency of the airfoil [3,6,7]. Variation of the maximum height of roughness can also play an import role to improve the aerodynamics characteristics of the airfoil, the ratio of maximum height to boundary layer thickness should not greater than one i.e. $\left(\frac{k}{\delta} > 1\right)$[1].

2. FLOW CONTROL APPROACH
Boundary layer behavior can be altered variation of following parameter such as (a) Reynolds number (b) Angle of Attack (c) geometry of roughness (d) roughness placement (e) height of roughness (f) number of roughness. In this research work, study has been performed on (a) to (d). In present study E, 398 airfoils has been selected for study because it is one of the low Reynolds number candidates. In this research work selected method for flow control is introducing roughness at upper surface of airfoil near leading edge, to investigate the boundary layer effect over stthe andard E398 airfoil. For this purpose standard E398 airfoil with chord length 0.128m, modified with single bump centered at 10%C, maximum height K = 0.9855%C and width w = 6.645%C, also location of bump changes to 5%C, 15% 20%C and 30%C Numerical Analysis has been carried out on all models at Re_C 18000 to 54000, for further details of experimental work found in [1].

Fig. 1: Smooth E398 airfoil   **Fig. 2: Uni-bump E398 airfoil**

3. GRID GENERATION
A C-grid was used for all computations; the upper and lower far-field boundaries are 10 times of chord length from the airfoil, on the other hand upstream and downstream are 8 and 10 chord respectively. A structured mesh with the quadrilateral cell was generated.

4. MATHEMATICAL MODEL
In this study flow over airfoil is considered as an incompressible flow, the flow having self-sustaining fluctuations of flow properties, the flow is known as turbulent flow. To describe the properties like mass, momentum, and energy, from the continuity equation, predict the conservation mass and for momentum and energy, Navier stock equation is used to predict the properties of flow [10]

$$\frac{\partial \rho}{\partial t} + \frac{\partial(\rho u)}{\partial x} + \frac{\partial(\rho v)}{\partial y} + \frac{\partial(\rho w)}{\partial z} \tag{1}$$

$$\rho\left[\frac{\partial u}{\partial t} + u\frac{\partial u}{\partial x} + v\frac{\partial u}{\partial y} + w\frac{\partial u}{\partial z}\right] = -\frac{\partial p}{\partial x} + \mu\left[\frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2} + \frac{\partial^2 u}{\partial z^2}\right] \tag{2}$$

$$\rho\left[\frac{\partial v}{\partial t} + u\frac{\partial v}{\partial x} + v\frac{\partial v}{\partial y} + w\frac{\partial v}{\partial z}\right] = -\frac{\partial p}{\partial y} + \mu\left[\frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2} + \frac{\partial^2 v}{\partial z^2}\right] \tag{3}$$

$$\rho\left[\frac{\partial w}{\partial t} + u\frac{\partial w}{\partial x} + v\frac{\partial w}{\partial y} + w\frac{\partial w}{\partial z}\right] = -\frac{\partial p}{\partial y} + \mu\left[\frac{\partial^2 w}{\partial x^2} + \frac{\partial^2 w}{\partial y^2} + \frac{\partial^2 w}{\partial z^2}\right] \tag{4}$$

A most common form of their practice

$$\rho\frac{Du}{Dt} = X - \frac{\partial p}{\partial x} + \frac{\partial}{\partial x}\left[\mu\left(2\frac{\partial u}{\partial x} - \frac{2}{3}\text{div}*w\right)\right] + \frac{\partial}{\partial y}\left[\mu\left(\frac{\partial u}{\partial y} + \frac{\partial v}{\partial x}\right) + \frac{\partial}{\partial z}\left[\mu\left(\frac{\partial w}{\partial x} + \frac{\partial u}{\partial z}\right)\right]\right] \tag{5}$$

$$\rho\frac{Dv}{Dt} = Y - \frac{\partial p}{\partial x} + \frac{\partial}{\partial y}\left[\mu\left(2\frac{\partial v}{\partial y} - \frac{2}{3}\text{div}*w\right)\right] + \frac{\partial}{\partial z}\left[\mu\left(\frac{\partial v}{\partial z} + \frac{\partial w}{\partial y}\right) + \frac{\partial}{\partial x}\left[\mu\left(\frac{\partial u}{\partial y} + \frac{\partial v}{\partial x}\right)\right]\right] \tag{6}$$

$$\rho\frac{Dw}{Dt} = Z - \frac{\partial p}{\partial x} + \frac{\partial}{\partial z}\left[\mu\left(2\frac{\partial w}{\partial z} - \frac{2}{3}\text{div}*w\right)\right] + \frac{\partial}{\partial y}\left[\mu\left(\frac{\partial w}{\partial x} + \frac{\partial u}{\partial z}\right) + \frac{\partial}{\partial y}\left[\mu\left(\frac{\partial v}{\partial z} + \frac{\partial w}{\partial y}\right)\right]\right] \tag{7}$$

4.1 Turbulent model
The turbulent model consists of semi-empirical equation related to main flow variables, constants used in the equation were found through experimental investigation. There is no single model their performance is universally accepted for all problems. In this numerical study, Reynolds averaged Navier-Stokes (RANS) approach based K-epsilon model was used to understand the flow characteristic over E398 airfoil.

4.2 $K-\in$ Turbulent model
This model was given by Jones and Launder, it is known as standard model for simulation of turbulent flow Modified Boussinesq eddy viscosity model overcomes the mixing length, turbulent viscosity is not defined where value of shear is zero $\left(\frac{\partial u}{\partial y} = 0\right)$, to add turbulent viscosity with Reynolds stresses.

$$V_t = \sqrt{K} \tag{8}$$

$$\mu_t = C_p l_m k^{1/2} \tag{9}$$

In this model two new transport variables, k and ϵ are introduced.

4.3 The Realizable $K-\in$

Three different $K-\in$ model are available such as standard $K-\in$, RNG- $K-\in$, realizable $K-\in$ model. Realizable $K-\in$ model performs better than the standard $K-\in$, when fluid flow against positive pressure gradient and flow separation occurs over solid surfaces. Because of realizable mothe del contains the new formulation for turbulent viscosity and introduces other transport equation for the dissipation rate($\in$). Some mathematical constrain on stress satisfy by this model that's why this model is realizable. In this numerical study realizable $K, -\in$ model is used because of fluid flow over airfoil, because fluid flow against the strong adverse pressure and flow separation occurs over the surface of airfoil. [10]

Transport equation of K and $\in$ for all three model standard $K-\in$, RNG $K-\in$ model and Realizable $K-\in$ model are similar, only differ in calculating turbulent viscosity (μ_t) and Prandtl number. Transport equations for K and $\in$ are as under:

$$\frac{\partial(\rho k)}{\partial t} + \frac{\partial(\rho k u_j)}{\partial x_i} = \frac{\partial}{\partial x_i}\left[\left(\mu + \frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right] + G_k + G_b - \rho\epsilon - Y_M + S_k \tag{10}$$

and,

$$\frac{\partial(\rho\epsilon)}{\partial t} + \frac{\partial(\rho\epsilon u_j)}{\partial x_j} = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_\epsilon}\right)\frac{\partial\epsilon}{\partial x_j}\right] + \rho C_1 S_\epsilon - \rho C_2 \frac{\epsilon^2}{K+\sqrt{v\epsilon}} + C_{1\epsilon}\frac{\epsilon}{K}C_{3\epsilon}G_b + S_\epsilon \tag{11}$$

Where,

$$C_1 = max\left[0.43, \frac{\eta}{\eta+5}\right]$$

$$\therefore \eta = S\frac{K}{\epsilon}$$

Eddy viscosity computed from following formula

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

The default model constants are

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

The pressure based unsteady solver has been used in simulation with Ansys Fluent 14.0. The realizable k$-\epsilon$ model was used with higher order discretization. Simplec scheme is used for pressure velocity coupling. Second order upwind discretization is used for pressure momentum, turbulent kinetic energy and turbulent dissipation, first order upwind is used for transient formulation. Defaults values of under relaxation factors are used.

In this study, Simulation has been carried out at Reynolds number 18,000, 36,000 and 54,000 for the various angles of attack ranging from 0° to 15°. Different inlet velocity specified for different Reynolds number, for Re 18,000, 36,000 and 54,000 the inlet velocities are 2.109 m/sec, 4.218 m/sec and 6.327 m/sec respectively.

5. RESULTS AND DISCUSSION

5.1 Flow Visualization

Streamlines give us a general idea about the flow field near the region of the airfoil such as separation, recirculation, and reattachment of fluid. Streamlines for both smooth airfoil and uni-bump airfoil are the plot for Re 18000 at AoA 11°. Fluid separates from solid surfaces at 54% C over a smooth airfoil, but there are no significant improvements with the addition of bump at 10%C.
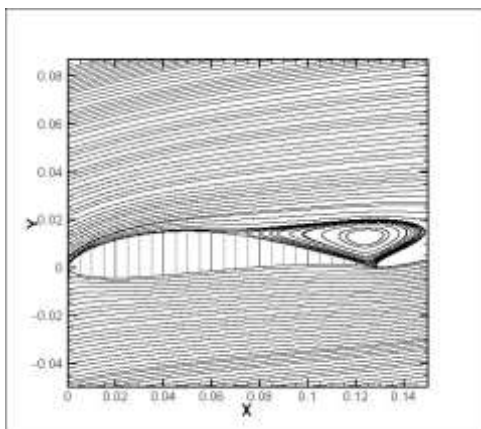

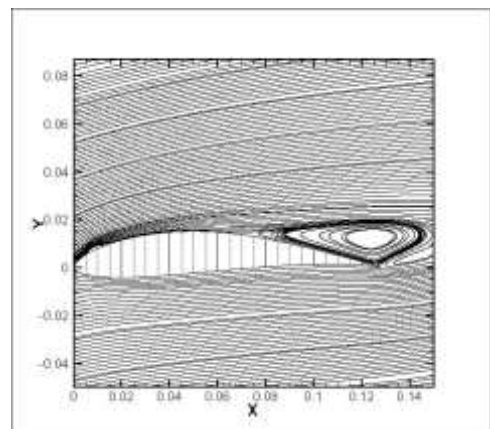**Fig. 3: Streamline of Smooth E398, Re 18000,**

Fig. 4: Streamline of Uni-bump E398, Re 18000, AoA 11°

In case of Reynolds number 36000 and Angle of attack 11°, see figure 5 and 6 it is observed that separation starts from 46%C for the smooth airfoil, the addition of bump delay separation for approximately 10%C, also the addition of bump reduces the size of vertices.

When the center of bump changes from 10%C to 5%C from leading edge then separation starts from 58%C and when single bump at 15%C, 20%C and 30%C, separation starts from 54%C, 50%C and 46%C respectively. It is observed that, when a bump is near leading edge, it is more effective to delay flow separation, as bump shifted toward the maximum thickness then separation point moves towards leading edge. It increases the separated flow region causes to decrease the suction pressure over the upper surface

of an airfoil leads to decreases the difference of pressure between upper and lower surfaces decreases the aerodynamic efficiency of the airfoil.
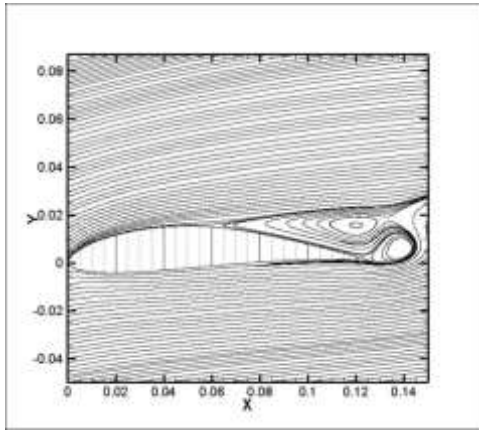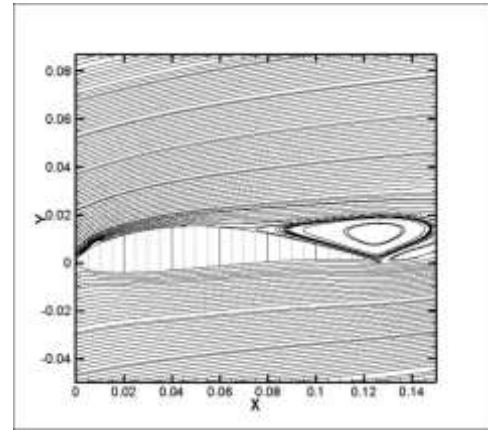

Fig. 5: Streamlines of smoothE398, Re 3600, AoA 11º

Fig. 6: Streamlines of smoothE398, Re 3600, AoA 11º

When Reynolds number increases to 54000, streamlines plotted at an angle of attack 11°. From figure 7 and 8, It is observed that separation start form 58%C for both smooth and uni-bump airfoil, there is no significant change separation, at higher Reynolds number there is no significant change in performance of airfoil as compare to Reynolds number 18000, and 36000.
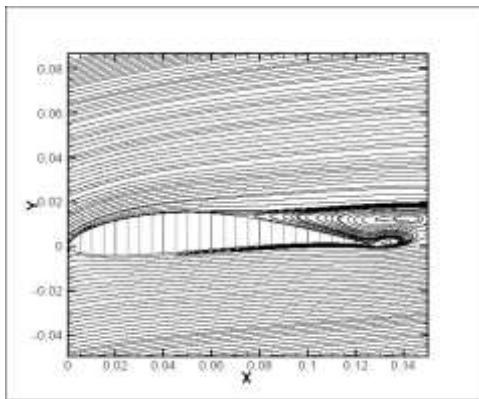
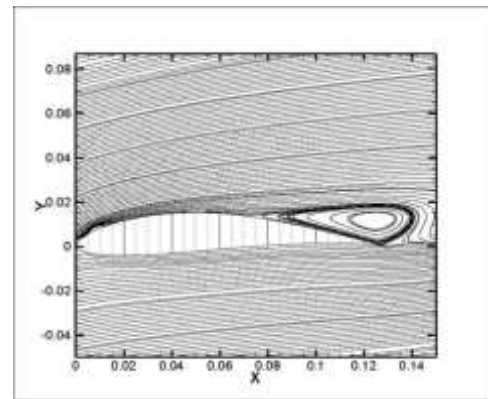**Fig. 7: Streamlines of smoothE398, Re 54000, AoA 11º**

Fig. 8: Streamlines of smoothE398, Re 54000, AoA 11º

5.2 Aerodynamic Performance
Performance of an airfoil is measured in terms of lift force and drag force. An efficient airfoil produces higher lift force and lower drag force. From figure 09, 10 and 11, when the angle of attack is in between 7° and 15°, C_L and aerodynamic efficiency of the uni-bump airfoil is higher than the smooth airfoil 10% and 17% respectively and C_D of the smooth airfoil is 6% higher than the uni-bump airfoil.

From figure 12 to 14, For Reynolds number 36000, when the angle of attack is between 0° and 11° Co-efficient of lift generated by a uni-bump airfoil is 13% greater than the smooth airfoil and coefficient of drag of uni-bump airfoil is 5% higher than the smooth airfoil at an angle of attack 0° to 7°. The aerodynamic performance with the addition of bump is higher 11% higher than the smooth airfoil at an angle of attack 0° to 11°. As Reynolds number increases to 54000 aerodynamic efficiencies of the uni-bump airfoil is 6% less as compared to the smooth airfoil. At higher Reynolds number addition of bump over the surface of airfoil near leading edge decrease lift force and increases drag force, because the magnitude of the difference of pressure between upper and lower surface of the uni-bump airfoil is less than the smooth airfoil. The maximum velocity of uni-bump is less as compared to smooth airfoil which reduces the dynamic pressure over an airfoil which decreases the difference of pressure; it leads to decreases lift force produced by Airfoil.
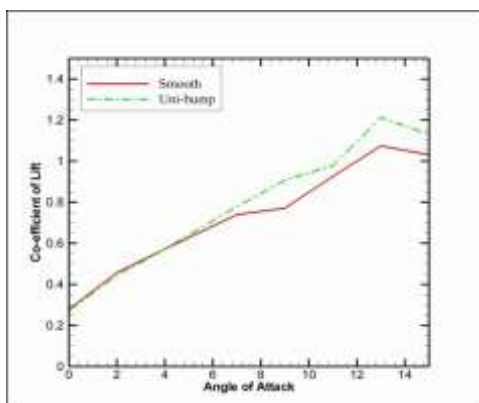

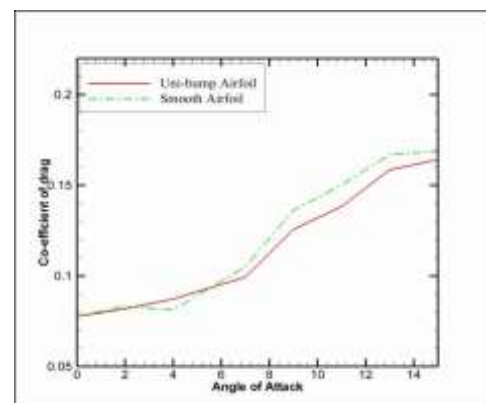**Fig. 9: Co-Efficient of Lift, Re 18000**
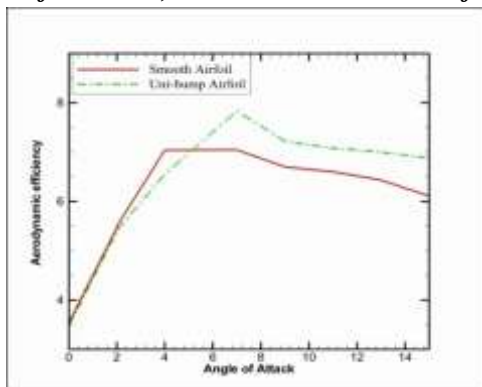

Fig. 10: Co-Efficient of Drag, Re 18000

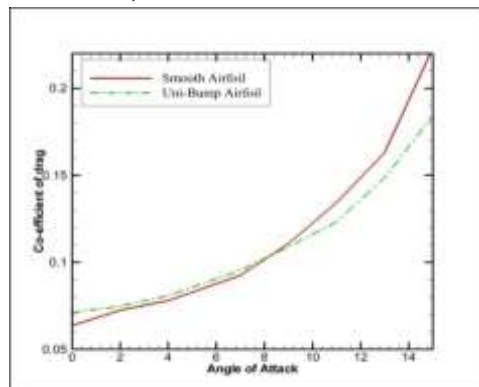**Fig. 11: Aerodynamic Efficiency, Re 18000**

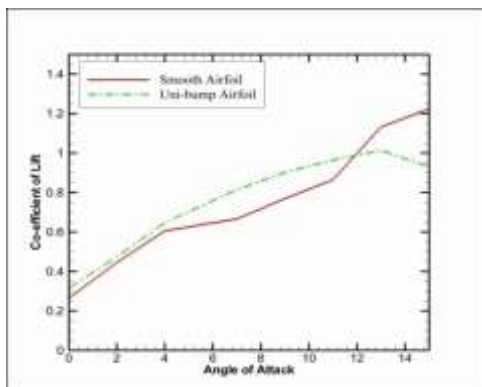**Fig. 12: Co-Efficient of Drag, Re 36000**

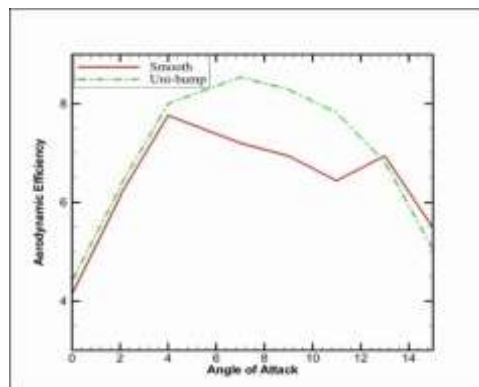**Fig. 13: Co-Efficient of lift, Re 36000**

Fig. 14: Aerodynamic Efficiency, Re 36000
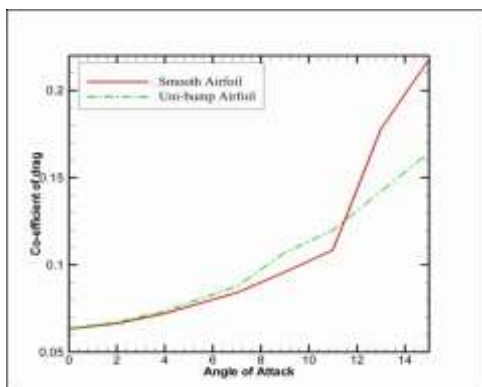

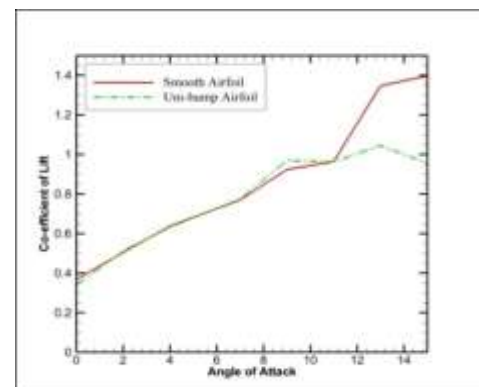**Fig. 15: Co-Efficient of drag, Re 54000**
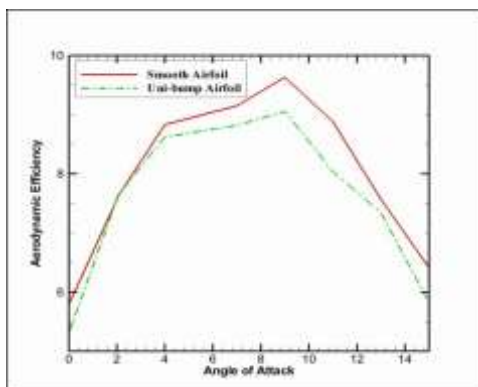

Fig. 16: Re 54000Co- Efficient of lift

Fig. 17: Aerodynamic Efficiency Re 54000

In this research work analysis has been performed for modified airfoil by changing the position of a single bump from 5%C to 30%C from leading edge, results were compared with smooth airfoil as well as with modified airfoil, to find out the best location of a bump over the surface of an airfoil. Results were compared at Reynolds number 36000 and Angle of attack 7^0 and 11^0. From table 0, observed that the performance of a smooth airfoil is lower as compared to the uni-bump airfoil. When the bump is placed at 5%C, aerodynamic efficiency of the uni-bump airfoil is higher than a smooth airfoil. However, the performance of uni-bump airfoil is decreased as bump shifted from 5%C towards the maximum thickness of airfoil as compare to airfoil with a bump at 5%C.

Table 1: C_L, C_D and C_L/C_D of an airfoil with a bump at a different location, Re 36000

	Angle of Attack	C_L	C_d	C_L/C_d
Smooth	7°	0.6664	0.0925	7.2029
	11°	0.8647	0.1342	6.4397
Centre of bump 5%C	7°	0.8298	0.0940	8.8276
	11°	0.9647	0.1228	7.8519
Centre of bump 10%C	7°	0.8145	0.0953	8.5400
	11°	0.9645	0.1233	7.8193
Centre of bump 15%C	7°	0.7757	0.0943	8.2245
	11°	0.9270	0.1251	7.4082
Centre of bump 20%C	7°	0.7599	0.0939	8.0890
	11°	0.9022	0.1289	6.9997
Centre of bump 30%C	7°	0.7514	0.0999	7.5182
	11°	0.8922	0.1389	6.4226

6. CONCLUSION

Aerodynamic efficiency of an airfoil is improved with the addition of bump at 10%C from the leading edge of an airfoil. When Reynolds number is 18000, C_L is 10% higher, C_D 6% less than smooth airfoil, the angle of attack is between 7^0 and 15^0 and at a low angle of attack aerodynamic performance of uni-bump airfoil is less as compared to the smooth airfoil. As Reynolds number increases to 36000, C_L is 13% greater than the smooth airfoil and aerodynamic efficiency is 11% higher than to smooth airfoil for the moderate angle of attack. At Reynolds number 36000 analyses performed, by changing the position of a bump from 5%C to 30%, Co-efficient of lift and aerodynamic efficiency decreases and coefficient of drag increases as bump moves toward maximum thickness of an airfoil. For Reynolds, number 54000 aerodynamic efficiency of the airfoil is 6% less with the addition of bump as compared to the smooth airfoil. It is concluded that the addition of a single bump increases the performance of E398 airfoil at Reynolds number 18000 and 36000 when a bump is placed at 5%C and 10%C from leading edge. When the bump is placed at near to maximum thickness of an airfoil shows the less improvement in aerodynamic performance of an airfoil as compare to the bump at 5%C and 10%C.

7. REFERENCES

[1] Arvind Santhanakrishnan. , Jamey D. Jacob, Effect of Regular Surface Perturbations on Flow Over an Airfoil", 35th AIAA Fluid Dynamics Conference and Exhibit, 6 - 9 June 2005, AIAA 2005-5145.
[2] Daniel A. Reasor, Raymond P. LeBeau, Suzanne Weaver Smith, Jamey D. Jacob "Flight Testing and Simulation of a Mars Aircraft Design Using Inflatable Wings", 45th AIAA Aerospace Sciences Meeting and Exhibit, 8 - 11 January 2007, AIAA 2007-243.
[3] GAD-EL-HAK, Mohammad, "flow control; Passive, Active, And Reactive Flow Management", 2000.
[4] A. Abbas, J. de Vicente, E. Valero, "Aerodynamic Technologies to Improve aircraft performance", Aerospace Science and Technology 28 (2013), , PP 100-132.
[5] Md. Amzad Hossain, Md. Nizam Uddin, Md. Rasedul Islam, Mohammad Mashud, "Enhancement of Aerodynamic Properties of an Airfoil by Co-Flow Jet (CFJ) Flow", American Journal of Engineering Research (AJER), Volume-4, Issue-1, pp-103-112.
[6] Deepanshu Srivastav, "Flow Control Over Airfoils Using Different Shaped Dimples", 2012 International Conference on Fluid Dynamics and Thermodynamics Technologies (FDTT 2012), IPCSIT vol. 33(2012).
[7] Md. Abdullah Al-Bari, Mohammad Mashud, 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, ISSN 1819-6608.
[8] A. Dhiliban, P.Meena, P.S. Narasimhan, M. Vivek, S. Nadaraja Pillai, K.M Parammasivam, "Aerodynamic Performance of Rear Roughness Aerofoils", The Eighth Asia Pacific conference on Wind Engineering, December 10-14, 2013.
[9] Hyoung Woo OH, "Computational Fluid Dynamics", Intech, ISBN 978-953-7619-59-6, 2013
[10] C. Potter., C. Wiggert, H. Ramadan, "Mechanics of Fluids", 4th edition, Global Engineering, 2011.
[11] Yunus A. Çengel, "Fluid mechanics", 3rd edition, McGraw-Hill, 2013.
[12] Raymond P. LeBeau, Jr., Trey D. Gilliam, Andrew Schloemer, "Numerical Comparison of Flow over Bumpy Inflatable Airfoils", 47th AIAA Aerospace Science Meeting and Exhibit, 5-8 January 2009, AIAA 2009-1306.
[13] David Takahashi., Raymond P. LeBeau, Jr., "Computational Investigation of Flow over Inflatable Airfoils at Multiple Reynolds Number", 49th AIAA Aerospace Sciences Meeting including the New Horizons Forum and Aerospace Exposition, 4 - 7 January 2011, AIAA 2011-377.
[14] Thomas Hauser., Todd Alexander Johansen., Raymond P. LeBeau, Jr. "Computational Optimization of a Low Reynolds Number Inflatable Airfoil",20th AIAA Computational Fluid Dynamics Conference, 27 - 30 June 2011, AIAA 2011-3534.
[15] Douvi c. Eleni, Tsavalos I. Athanasios, Margaris P. Dionissios. "Evaluation of the turbulence for the simulation of the flow over a National Advisory Committee for Aeronautics (NACA) 0012 airfoil", Journal of Mechanical Engineering Research vol. 4(3) pp 100-111, 2012.
[16] Daniel A. Reasor Jr., Raymond P. LeBean Jr. " Numerical Study of Bumpy Airfoil Flow Control For Low Reynolds Number", 37th AIAA Fluid Dynamics Conference, June 25-28 2007, AIAA 2007-4100.
[17] Syed Hasib Akhter Faruqui, Md. Abdullah Al-Bari, Md. Emran, Ahsan Ferdaus., "Numerical Analysis of the role of the bumpy surface to control the flow separation of an airfoil",10th Internal Conference on Mechanical Engineering ICME 201, PP255-260.
[18] Usni, M, "Aerodynamics of a Low Reynolds Number Inflatable Wing," Master Thesis, University of Kentucky, 2004.

BIOGRAPHY

Asif Ali Laghari
Research scholar
Machanical Engineering
B.E 2012 (Quaid-e-Awam University College of Engineering & Technology, Pakistan)
M.E 2016 (National University Science Technology, Pakistan)

Dr. Shafiq-u-Rehman
Assistant professor
Machanical Engineering (National University Science Technolgy, Pakistan)
B. E 2008 (Mehran University of Engineering and Technology, Jamshoro)
PhD 2014 (University of Manchester, UK)

Dr. M. Tarique Bhatti
Assistant professor
Machanical Engineering (Quaid-e-Awam University College of Engineering & Technology, Pakistan)
B. E 1985 (Mehran UET, Pakistan)
PhD 1991 (Dublin City University, Ireland)

Allah Rakhio Golo
Research Fellow
Mechanical engineering
State Key Laboratory of Mechanical Transmission Chongqing university, Chongqing, china
B.E 2018 (Mehran UET, Pakistan).

Manthar Ali Khoso
B.E 2012 (Mehran UET, Pakistan)
M.E (Quaid-e-Awam University College of Engineering & Technology, Pakistan)

Sajjad Bhangwar
Lecturer
Machanical Engineering (Quaid-e-Awam University College of Engineering & Technology, Pakistan)
B.E 2012 (Quaid-e-Awam University College of Engineering & Technology, Pakistan)
M.E 2017 (Quaid-e-Awam University College of Engineering & Technology, Pakistan)