Sindh Univ. Res. Jour. (Sci. Ser.) Vol.49(2) 403-408 (2017)

SINDH UNIVERSITY RESEARCH JOURNAL (SCIENCE SERIES)

Aerodynamic Performance Analysis of Rough Rectangular Aircraft Wing for Subsonic Flow

I. ALI, S. SAMO, D. KUMAR*, M. ABRO, T. HUSSAIN

Department of Mechanical Engineering, Mehran UET, Jamshoro

Received 23rd January 2016 and Revised 17th February 2017

Abstract: Present research aims to analyze the effect of surface roughness on the aerodynamic performance of aircraft wing. Sand grain roughness was introduced over the wing surface through Computational Fluid Dynamics (CFD) tool. Surface roughness effect was reported at two different plane velocity one of them lies in the range of incompressible flow while other is in the range of compressible flow regime. Turbulences were modeled through $k-\varepsilon$ realizable and $k-\omega$ SST model for incompressible and compressible flows respectively. CFD simulation results estimated that drag increases and lift decreases with increase in roughness height for incompressible flow. In case of compressible flow, analysis results showed that increase in roughness height causes significant increment in drag along with some increase in lift force. On the basis of obtained results it is concluded that wing performance is highly sensitive to roughness along with that it is also analyzed that increase in roughness height causes decrease in lift to drag ratio.

Keywords: Surface roughness, Roughness height, Aerodynamic performance, Computational fluid dynamics, Sub-sonic flow

1. INTRODUCTION

The wing is one of major aircraft component that greatly affect its aerodynamic performance. Aircraft aerodynamic performance is highly dependents on drag performance. Drag performance is majorly affected by surface profile. Therefore special paints are used to paint over aircraft body in order to reduce the friction between air and wing structure (Salazar, 2013). Hence to enhance plane efficiency it became necessary to analyze the effect of various parameters that enhances drag production. Surface roughness is one of extreme important parameters that affect fluid flow behavior over the wing. In literature surface roughness is viewed from two separate perspectives one to enhance lift and other to avoid flow separation.

(Zhou, 2012) analyzed the effect of surface roughness over SD7003 rectangular aircraft wing through introducing small bumps of different specification over the wing surface at different locations. Results showed that surface roughness improve Lift to Drag (L/D) ratio and helps to avoid flow separation (Kumar, 2015). carried research to predict effect of riblets over wing performance and found that L/D ratio was improved through applying riblets over wing surface. (Santhanakrishnan, 2005). made attempt to predict effect f large scale surface roughness on the fluid flow behavior over airfoil, results showed that flow separation was delayed (Kerho, 1997). introduced roughness over 2D airfoil to visualize its effect on flow behavior. Their results showed that flow separation was decreased and it also prolongs transition in boundary layer (Hövelmann, *et al.*, 2016). carried out research to predict effect of surface roughness on flow behavior over the round leading-edge 53º sweep diamond wing. Result showed that flow behavior is highly sensitive to roughness height and leading edge contour (Saeed, *et al.*, 2014). predicted effect of surface roughness on the flow behavior over swept cross flow wing, found that roughness encourages transition in flow (Robert 2006). researched to analyze effect of roughness height, their spacing and skewness on the flow behavior. Results showed that flow behavior was observed to be highly sensitive to roughness height rather than spacing among them and skewness (Kurz, 2015). analyzed the influence of cylindrical roughness element on the transition location, and flow behavior over the 3D aircraft. Results found that transition location was shifted and advance transition of flow was observed.

(Hood, 1939) carried out series of wind tunnel tests to analyze effect rivets and surface roughness over the drag performance of aircraft wing. On the basis of wind tunnel tests Hood suggested a mathematical expression for calculation of drag. Later (Hood, 1938) estimated the effect of paint spray and carborundum grains on the drag production over wing for subsonic flow. Hood experimentally predicted that 30% and 63% increase in profile drag was achieved due spray of paint and carborundum grains respectively. (Srivastav, 2012). introduced roughness over both the wing surfaces and his research results found that by improving surface

++Corresponding author Email: tanweer.hussain@faculty.muet.edu.pk
*Department of Mechanical Engineering, Mehran UCET, Khairpur.

characteristics drag force decreases but lift force was also reduced (Dhiliban, 2013) predicted roughness effect on aerodynamic efficiency over NACA 0018 airfoil in subsonic flow conditions. His results showed that roughness on lower surface of wing was proved to be beneficial than on the upper surface of wing.

From extensive literature review it is observed that much of related research on the surface roughness is related to its effect on flow behavior. It was also observed that different roughness profile has different effect. Since sand grain roughness affect was not yet got much attention from researchers. Therefore present study estimated effect of sand grain roughness on aerodynamic performance of aircraft wing through widely adopted, reliable and cost computational Fluid Dynamics technique.

2. ROUGHNESS PARAMETERS

Fluid Flow around solid structures was observed in number of engineering application. When the fluid flows near the solid body fluid particles that are in contact with solid surface hiders at the solid structure due to adhesive forces and that condition is named as no slip condition. The surface profile of solid structure highly affects the flow behavior and transfer of other physical quantities such as heat. In case of wall bounded flow these effects should be taken into account to get realistic results. In FLUENT wall roughness effects can be easily modeled by defining two parameters named as Roughness Height K_s and the roughness constant C_s. For the smooth surfaces FLUENT by default assume the value of roughness constant as (C_s =0.5) and zero for roughness height. For uniform sand grain roughness, roughness height is exploited to define sand grain size. Present study is conducted through applying sand grain roughness therefore its roughness constant value remain same whereas roughness height value varies in range from 0.0 to 0.0007m.

3. NUMERICAL FEATURES

3.1 Governing Equation

The fluid flow governing equations like continuity equation, momentum equation and energy equations are derived through applying the basic conservation principal of mass, momentum as well as energy. Those equation are Partial Differential equations (PDE) which difficult to solve analytical or numerically because they take into account the small fluctuation in fluid properties. Therefore theses complex equations are simplified through applying Reynolds averaging. By using averaging velocity of fluid in X-direction is

$$u_i = \bar{u}_i + u_i'$$

Whereas $\bar{u}_i$ and u_i' represents average and fluctuating components of velocity respectively. Similarly, for any other scalar quantity

$$\rho = \bar{\rho}_i + \rho_i'$$

As in above expression density is decomposed into average and fluctuating components. By applying Reynolds averaging technique following equations are obtained in terms time averaged components.

$$\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x_i}(\rho u_i) = 0 \tag{1}$$

$$\frac{\partial}{\partial t}(\rho u_i) + \frac{\partial}{\partial x_j}(\rho u_i u_j) = \frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j}\left[\mu\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} + \frac{2}{3}\delta_{ij}\frac{\partial u_i}{\partial x_i}\right)\right] + \frac{\partial}{\partial x_j}\left(-\rho \overline{u_i' u_j'}\right) \tag{2}$$

The Eqs. (1) & (2) are known as Reynolds average Navier-Stokes equations. In Eq. (2) the last $\left(-\rho \overline{u_i' u_j'}\right)$ is known as Reynolds Stresses and appears to take into account the effect of turbulences generated in flow. In order to model this term FLUENT provides number turbulence models. The turbulence modeling in flow is one critical steps and its proper modeling is extremely important to reduce numerical errors. In present study simulation is performed on different plane velocity out which one lies in incompressible flow regime and other in compressible flow regime. Hence k-ε realizable model is used for modeling of incompressible flow, because it is widely adopted for turbulent flows but it doesn't provides good results in cases of high speed compressible flows and flow having large pressure variation (Wilcox, 1998). Hence k-ω SST model is used for compressible flow. Because it has ability to analyze flow behavior inside boundary layer and also flow separation regions (Menter, 1992). The governng transport equations of k-ε realizable and SST k-ω model are given below.

$$\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_i}(\rho k u_i) = \frac{\partial}{\partial x_j}[(\mu + \frac{\mu_t}{\sigma_k})\frac{\partial k}{\partial x_j}] + G_k + G_b - \rho\varepsilon - Y_M + S_k \tag{3}$$

$$\frac{\partial}{\partial t}(\rho\varepsilon) + \frac{\partial}{\partial x_i}(\rho\varepsilon u_i) = \frac{\partial}{\partial x_j}[(\mu + \frac{\mu_t}{\sigma_\varepsilon})\frac{\partial \varepsilon}{\partial x_j}] + C_{1\varepsilon}\frac{\varepsilon}{k}(G_k + C_{3\varepsilon}G_b) - C_{2\varepsilon}\rho\frac{\varepsilon^2}{k} + S_\varepsilon \tag{4}$$

$$\frac{\partial(\rho k)}{\partial t} + \frac{\partial(\rho k u_i)}{\partial x_i} = \frac{\partial}{\partial x_j}(\Gamma k \frac{\partial k}{\partial x_j}) + \tilde{G}_k - Y_k + S_k \tag{5}$$

$$\frac{\partial(\rho\omega)}{\partial t} + \frac{\partial(\rho\omega u_i)}{\partial x_i} = \frac{\partial}{\partial x_j}(\Gamma\omega\frac{\partial \omega}{\partial x_j}) + \tilde{G}_\omega - Y_\omega + D_\omega + S_\omega \tag{6}$$

3.2 Geometry and Gird generation

In order to analyze the effect of surfcae roughness on the aerdoynamic performance of aircraft wingNACA2412 airfoil was selected whose coordinates were taken fron National Advisory space website.Those coordinates points was imported to Pro-Engineer software where the 3D mdel of rectangular aircfat wing was developed. Designed wing has chord of 150mm and span of 300mm. Designed 3D model of wing was imported in ANSYS design Modeler to create fluid domain around it. Structural wing volume is subtrated from fluid domain volume and futher preocessed. The next and most important step in CFD simulation is gid generation in order to solve all the fluid flow govenig equation on each control volume.Tetrahedral unstructured mesh was generated by applying patch conforming algorithm to get refined mesh at point of high pressure gradient. In **(Table-1)** mesh details are provided along with type of element.

Table1: Showing fluid domain mesh details

Domain	orthogonal quality	Nodes	Elements	Tetrahedral
Fluid	**0.86**	**52031**	**282639**	**282639**

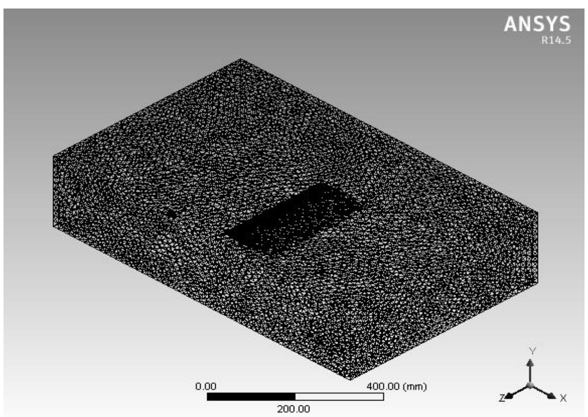

Fig 1: Showing the Complete Meshed Domain along with wing.

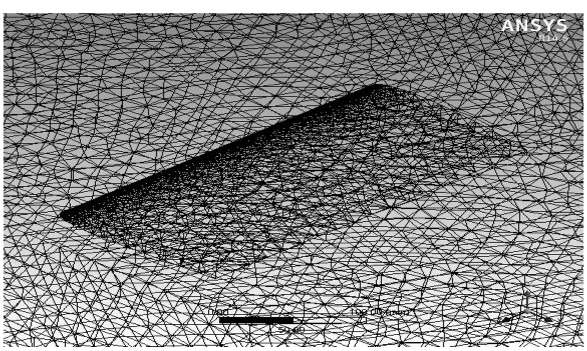

Fig 2: Zoomed View near the wing is shown

3.3 Simulation setup

In present study flow governing RANS equations were solved through finite volume unstructured tetrahedral mesh. Since study was conducted on two different aircraft velocities which lie in incompressible as well as compressible flow regimes. Pressure-velocity coupling was selected as SIMPLE and two different turbulent models were used for incompressible and compressible flow. FLUENT provides series of turbulent models according to the flow physics. Turbulence model selection was done on the basis of extensive literature review. In modeling incompressible flow at the plane velocity of 80m/s steady state simulation was done through utilizing pressure based solver and turbulences are modeled by K-ε viscous Realizable model. Surface roughness was introduced over wing by considering it as rough wall and setting values surface roughness height (K_s). in second case fluid flow over the wing surface is modeled by considering the effects of compressibility because in this case plane mach number is greater than 0.3 hence flow is considered as compressible. In order to take into account the effect of compressibility density based solver has to be used because it provides more accurate results than pressure based solver.

(Chen, 2010), (Veress, 2012), (Zheng, *et al.*, (2012) In this case steady state simulation was conducted and turbulences were modeled through k-omega SST viscous model. Energy model was also activated for encountering the effects of fluid heat generation and transfer. For modeling of compressible flows total properties at the inlet are need to calculate in order to reduce numerical error and to get more realistic results. Static properties were determined through below mentioned equations.

$$\frac{p_o}{p} = \left[1+\left(\frac{\gamma-1}{2}\right)M^2\right]^{\frac{\gamma}{\gamma-1}} \quad \textbf{(7)}$$

$$\frac{T_o}{T} = \left[1+\left(\frac{\gamma-1}{2}\right)M^2\right] \quad \textbf{(8)}$$

Whereas in Eqs.(7) & (8) p_o, p, γ, T_o, T and M represent total pressure, static pressure, adiabatic index for air, total temperature, static temperature and Mach number respectively. By using Eqs.(7) & (8) the values of total temperature and pressure were calculated for M=0.7, γ =1.4 for air and of total pressure, temperature. Values obtained from these equations are p= 73048 Pa and T= 283 .24 K.

For compressible flow viscosity was calculated through applying ‘Sutherland’ law. Aerodynamic forces can be obtained through given formulas.

$$L = C_L \frac{\rho_\infty V_\infty{}^2 S}{2} \quad \textbf{(9)}$$

$$D = C_D \frac{\rho_\infty V_\infty{}^2 S}{2} \quad \textbf{(10)}$$

In this equation C_L, C_D, ρ_∞ and S are used to represent lift coefficient, drag coefficient, fluid density and wing swept area.

4. RESULT AND DISCUSSION

In present study CFD analysis of aircraft wing was performed at two different velocities of aircraft and at different values of surface roughness height. In order to analyze the variation of fluid parameters pressure, velocity, turbulence intensity and walls shear stress contours are given below. In **(Fig-3)** velocity contour over the aircraft wing was presented and from the velocity contour it was analyzed fluid velocity is zero at wing surface and increases at the upper surface of wing whereas fluid velocity decreases on the lower side of aircraft wing by satisfying Bernoulli'theorm. In Fig.4 pressure contour over the wing surface was presented and through visualizing flow behavior and variation in fluid parameters it was concluded that pressure decreases over the upper surface of wing and increases on the wing lower sides results production of lift force.

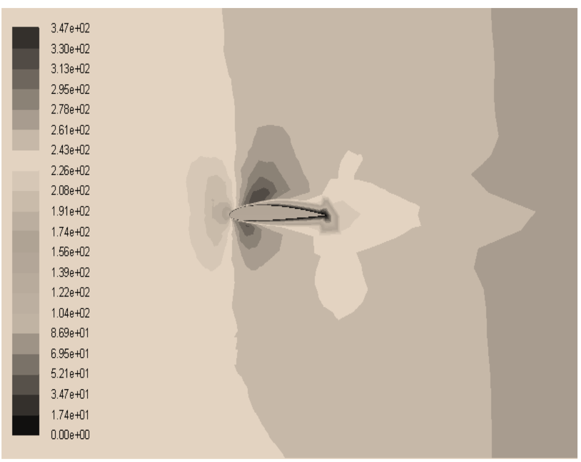

Fig. 3 Velocity contour near wing m/s

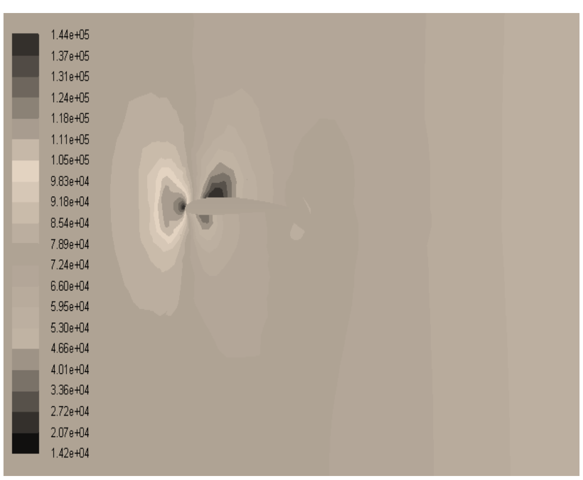

Fig 4. Static Pressure contour of near the wing in Pascal(Pa).

Turbulent intensity over the wing surface was presented in **(Fig-5)**. Through visual inspection of the turbulent intensity in was observed that it increase near the trailing edge of the wing and achieved highest value just behind the wing trailing edge. In **(Fig-6)** wall shear stress contour was presented and through visualizing wall shear stress contour it is concluded that wall shear stress increases near trailing edge and also keep on increasing just behind trailing edge.

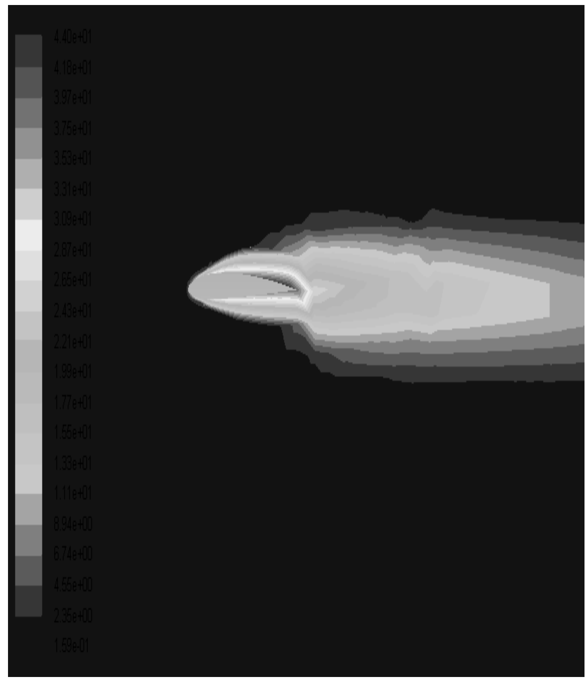

Fig:5 Turbulent intensity contour near the wing in %.

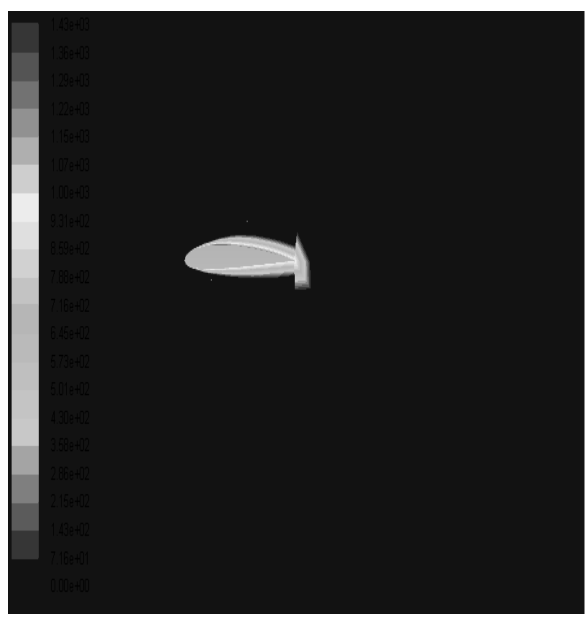

Fig: 6 Contour shows variation in wall shear stresses near aircraft wing in Pascal.

Table 2 Show variation L/D ratio at various values of roughness height

Velocity of air m/s	Roughness height	Roughness constant	Lift force (N)	Drag force (N)	Lift/Drag Ratio
80	0.0003	0.5	21.52	4.26	5.051643
80	0.0005	0.5	21.31	4.85	4.393814
80	0.0007	0.5	21.19	5.23	4.051625
80	0.0009	0.5	20.81	6.68	3.115269
80	0.0012	0.5	20.39	7.89	2.584284
242	0.0003	0.5	90.18	187.15	0.481859
242	0.0005	0.5	92.12	190.67	0.483138
242	0.0007	0.5	93.27	193.07	0.483089
242	0.0009	0.5	94.06	194.81	0.482829
242	0.0012	0.5	94.78	196.36	0.482685

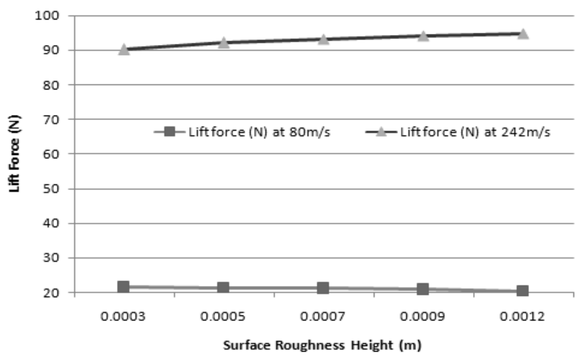

Fig. 7 Shows the variation of lift force against different values of roughness height

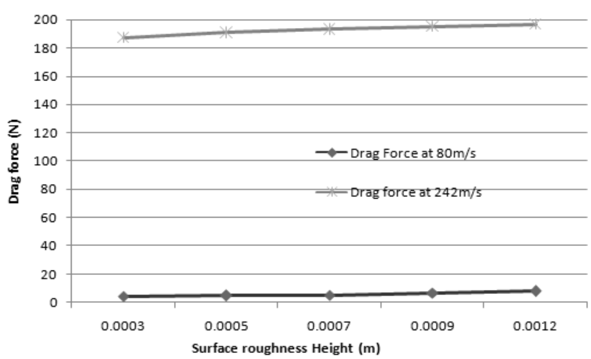

Fig. 8 Shows the variation of drag force against different values of roughness height

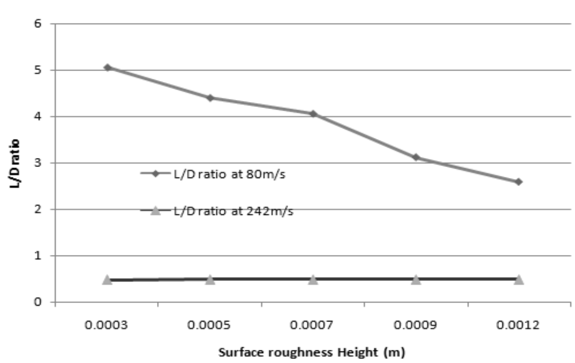

Fig. 9 Shows the variation of L/D ratio against different values of roughness height

In **(Fig-7)** variation of lift force against roughness height was presented at two different aircraft velocities. From graph it was analyzed that at 80m/s lift force decreases with increase in roughness height while at 242m/s lift increases with roughness height. In **(Fig-8)** variation of drag force against roughness height was presented and it was observed that drag increases in both the cases. From Fig. 9 it was concluded that there was significant decrease in L/D ratio in cases of incompressible flow whereas there was minute decrease in L/D ratio was observed for compressible flow.

5. CONCLUSION

In present study surface roughness effect was analyzed on the aerodynamic performance of aircraft wing. Computational Fluid Dynamics tool was used to simulate the effect of sand grain roughness the wing surface. Surface roughness effect was reported at two different plane velocity one of them lies in the range of incompressible flow while in second case flow was considered as compressible. CFD simulation results estimated that drag increases and lift decreases with increase in roughness height for incompressible flow. In case of compressible flow, analysis results showed that increase in roughness height causes significant increment in drag along with some increase in lift force. On the basis of obtained simulation results it is concluded that uniform sand grain roughness imposed no favorable effects on wing aerodynamic performance.

REFERENCES:

Chen, Z., A. Przekwas, (2010). A coupled pressure-based computational method for incompressible/compressible flows. Journal of Computational Physics, 229(24): 9150-9165.

Dhiliban, A., (2013). Aerodynamic Performance Of Rear Roughness Aerofoils. in The Eighth Asia-Pacific Conference On Wind Engineering Chennai, India.

Hood, M. J., (1938). The Effect of Surface Irregularities on Wing Drag. 3; Roughness.

Hood, M. J., (1939). Surface Roughness and Wing Drag: An American Opinion upon a Problem Becoming of Ever-Increasing Importance. Aircraft Engineering and Aerospace Technology, 11(9): 342-344.

Hövelmann, A., F. Knoth, C. Breitsamter, (2016). AVT-183 diamond wing flow field characteristics Part 1: Varying leading-edge roughness and the effects on flow Separation Onset. Aerospace Sci. and Technology.

Kerho, M. F., M. B. Bragg, (1997). Airfoil boundary-layer development and transition with large leading-edge roughness. AIAA journal, 35(1): 75-84.

Kumar, K. R., P. Maniiarasan, (2015). Reduction of skin friction drag by employing riblets. International Jour. of Engineering Research Technology.4(7) 64-68.

Kurz, H. B., M. J. Kloker, (2015). Discrete-roughness effects in a three-dimensional boundary layer on an airfoil by means of DNS. Procedia IUTAM, 14: 163-172.

Menter, F. R. (1992). Performance of popular turbulence model for attached and separated adverse pressure gradient flows. AIAA journal, 30(8): 2066-2072.

Roberts, S. K., M. I. Yaras, (2006). Effects of surface-roughness geometry on separation-bubble transition. Journal of turbomachinery, 128(2): 349-356.

Saeed, T., J. Morrison, M. Mughal, (2014). Roughness effects on swept-wing crossflow transition in moderate free-stream turbulence. in 29th Congress of International Council of the Aerospace Sciences.

Salazar, F., A. Barrientos, (2013). Surface roughness measurement on a wing aircraft by speckle correlation. Sensors, 13(9): 11772-11781.

Santhanakrishnan, A., J. D. Jacob, (2005). Effect of regular surface perturbations on flow over an airfoil. AIAA, 51 45-50

Srivastav, D., (2012). Flow control over airfoils using different shaped dimples. International Proceedings of Computer Science & Information Technology 33: 92-8.

Wilcox, D. C., (1998). Turbulence modeling for CFD. Vol. 2: DCW Industries La Canada, CA.

Veress, Á., J. Rohács, (2012). Application of Finite Volume Method in Fluid Dynamics and Inverse Design Based Optimization: INTECH Open Access Publisher.

Zheng, H. (2012). Computational fluid dynamics simulation of the supersonic steam ejector. Part 1: Comparative study of different equations of state. Proceedings of the Institution of Mechanical Engineers, Part C: Journal of Mechanical Engineering Science, 226(3): 709-714.

Zhou, Y., Z. Wang, (2012). Effects of surface roughness on separated and transitional flows over a wing. AIAA Journal, 50(3): 593-609.