Numerical Study of the Boundary Layer Separation Control on the NACA 0012 Airfoil using Triangular Rib

Dr. Mohammed Wahhab Al-Jibory 1, Hussain Abed Ali Shinan2

1, 2 Iraq, Kerbala University, Mechanical Engineering Department

1E-mail: Dr.Mohammad.wahab@uokerbala.edu.iq

2E-mail: huss1982ain@gmail.com

Abstract. This study includes a numerical simulation to investigate the aerodynamic behavior of the air flow around the surface of the NACA0012 airfoil, with and without a triangular rib. The triangular rib was used on the upper surface of the airfoil with size of 2% of the total chord length and variously located at 50, 70 and 90 percent of the chord length from the leading edge, separately as a passive control technique. Workbench-Fluent 17.2 version software was used to simulate the flow at 7 m/s freestream velocity (Re = 78,000), various attack angles (0˚ to 22˚) and chord length of 168 mm. The results indicated the presence of the triangular rib on the airfoil’s upper surface, at low angles of attack (less than 12˚), was a negative for the lift and drag results, at the angle of 12˚ it was a positive only for the location of 90%, while at the higher angles it was a positive for all locations and for all results where the lift increased and drag decreased. Therefore, the NACA0012 airfoil’s performance (lift to drag ratio) was significantly enhanced but, at 22˚, it had a very small effect for all results. It was concluded that the rib’s presence at 50% location delayed the stall angle by two degrees (from 14˚ to 16˚).

Keywords: NACA0012 Airfoil, Triangular Rib, Reynolds Number, Attack Angle, Lift, Drag, Airfoil Performance, Stall Angle
1. Introduction

The airfoil is the cross section of an aircraft wing or turbine blade in any engineering application that requires high lift and low drag, and designed with a geometry to achieve that purpose.

The boundary layer around the airfoil’s surface has a strong and direct effect upon its aerodynamic performance. Separation of that layer from the airfoil surface leads to large losses in performance and as the separation gets closer to the leading edge, these losses increase. Therefore, any improvement or degradation in the airfoil’s performance (lift increase, drag reduction, simultaneous lift increase and drag reduction or vice versa) means improvement or degradation in these applications’ performance.

Many and varied boundary layer separation control techniques have been used in the recent past to for prevent or delay the separation or reduce its area. These techniques can be classified into two main groups: active and passive. For the active techniques, an external energy is added to energize the stagnant boundary layer, for example, blowing or suction techniques. For the passive techniques no external energy is used, but the airfoil geometry is modified to energize that layer, for example the riblet or vortex generator techniques.

Some researchers have used both types of technique (active and passive) in the same work, for example using the active and passive vortex generators or using the step and blowing techniques together in the same research.

Huang et al. [1] used suction and blowing techniques to study performance of the NACA0012 airfoil, numerically. The study was conducted for three parameters, namely the location, velocity amplitude and angle of the jet at attack angle of 18°. The results showed that the optimum angle, location and amplitude of the jet to improve airfoil’s performance were 90°, 10%c from the leading edge and about 0.01 respectively for the suction and 90°, about 80%c from the leading edge and 0.2 respectively for the blowing technique.

The flow separation control on the NACA 0012 airfoil’s surface was investigated numerically by Hua Shan et al. [2] using both the active and passive vortex generators (VGs). The study was
conducted at attack angle of 6°, Re=100,000 and various dimensions of VG. It was concluded that the passive VGs reduced the separation region about 80%, but the active VG was more effective and the separation area was completely eliminated with that technique.

Performance of the NACA 0012 airfoil was investigated numerically at Re = 160,000 by YF Lin et al. [3], using sinewaves on both of the surfaces. It was concluded that at small attack angles (α<13°), there was slight reduction in the lift, whereas at the higher angles there was an improvement in the lift coefficient of about 20% compared with the original airfoil. Generally, with the sinewaves, the airfoil’s performance decreased at low angles of attack (less than 13°) but was enhanced at the higher angles.

The NACA 0018 airfoil’s characteristics were simulated numerically by Deepanshu Srivastav et al. [4] by application of outward and inward dimples, of different sizes and shapes, on the airfoil’s surface at 20 m/s air velocity and attack angles ranging from -5 to 15 degrees and with chord and span lengths of 16 and 0.8 cm respectively. It was concluded that, at the positive attack angles and with outward dimples, there was a reduction in drag compared with the conventional airfoil and the other cases.

The NACA 0012 airfoil’s performance was examined numerically by Kianoosh Yousefi et al. [5], both with and without perpendicular and tangential blowing at a freestream velocity of 7.3 m/s (Re=500,000), the chord at 100 cm and different attack angles of 12, 14, 16 and 18 degrees. The blowing slots were placed at 20%c from the trailing edge with different velocities (0.1, 0.3 and 0.5 percent of the freestream velocity) and length of 3.5%c. The results indicated that, at an 18-degree attack angle, 0.5% amplitude, use of the tangential blowing enhanced the airfoil’s performance by about 16%, while using the perpendicular blowing technique under the same conditions improved the performance by about 18% compared with the original case.

Syed Hasib et al. [6] conducted a numerical study to examine the NACA 4315’s performance without and with bumps on the airfoil’s surface at 80%c from the leading edge with height of 2.5%c (6.35mm). The results indicated that the air with the bumps separated from the surface at an attack angle of 15 deg. Without the bumps it separated at an attack angle of 9 degrees therefore, the airfoil’s performance improved with the bumps.
Aerodynamic efficiency of the NACA0012 airfoil was investigated numerically by Pritesh S Gugliya et al. [7]. The study was carried out at Reynolds number of \(3 \times 10^6\) and various angles of attack from -10 to 10 degrees. The results showed the maximum lift to drag ratio to be about 43 at an attack angle of 6 degrees.

Kanok Tongsawang [8] investigated experimentally the performance of the NACA 0015 airfoil at 6 m/s freestream velocity (Re=78,000) without and with trips (tubes of the carbon fiber) on the airfoil’s upper surface and with different diameters (6, 4, 3, 1.5 mm) at different locations of 10, 20, 30, 40 & 50\% of the chord length from the leading edge for wide range of attack angles (0° to 22°) and the test model with chord length of 20 cm and span of 70 cm. The results indicated that the optimum size and location of trips were 4 mm and 50\%c respectively and presence of the trips with the sizes of 1.5, 3 & 6 mm at all the locations for all the attack angles, except from 18° to 22°, reduced the lift and increased the drag.

Performance of the NACA 0015 airfoil without and with a suction slot was investigated by Ahmed S. Shehataa et al. [9]. They formed a suction slot from the lower to upper surface of the airfoil at location of 55\%c from the trailing edge with 1 mm diameter at conditions; 2.9 m/s flow speed and various attack angles. It was found that the suction slot enhanced the aerodynamic performance of the airfoil, delayed the stall angle and increased the coefficient of torque 17\% before the stall angle and about 40\% after the stall.

Characteristics of the NACA 0012 airfoil without and with dimples were studied numerically by Sandesh K. Rasal et al. [10]. That study was done by application of circular dimples at 30\%c from the trailing edge on the airfoil’s upper surface at 6 m/s air velocity, 30 cm chord length, a range of attack angles from 0 to 23 degrees and 3 mm (1\%c) dimple height. The results indicated that, at a 10-degree attack angle, the airfoil’s efficiency increased from 10 in the normal case of the airfoil to 12 in the modified airfoil with dimples.

The aerodynamic characteristics of the NACA0012 airfoil with and without a dimple were simulated numerically by Amit Kumar Saraf et al. [11]. That study was performed by use of a single inward dimple at different locations (75\%, 50\%, 25\% and 10\% of the chord length from the leading edge) on the airfoil’s upper surface, at a constant air speed of 7.3 m/s and various attack
angles. It was concluded that the dimple presence at location of 75% of the chord increased the lift by 7% and reduced the drag 3% compared with the smooth airfoil.

This research aims to improve the NACA 0012 airfoil’s performance (the lift to drag ratio) through increasing the lift, decreasing the drag or increasing the lift and decreasing the drag at same time, using a passive technique. That technique comprises the use of a triangular rib on the airfoil’s upper surface at different locations, 50, 70 and 90% of the chord length from the leading edge with size 2% of the chord length.

2. Numerical Approach

In this study, Workbench-Fluent 17.2 version software was used to investigate the NACA0012 airfoil’s aerodynamic characteristics (CL and CD) with and without the triangular rib over the upper surface of the airfoil and determining the lift to drag ratio (CL/CD) to show the effects of that technique on the airfoil’s aerodynamic performance.

CL and CD are the lift and drag coefficients, respectively.

In this work, the air flow around the airfoil surface could be assumed as steady, two-dimensional, viscous, turbulent and incompressible flow.

2.1 Wing Geometry

Before designing the geometry of test model (NACA 0012 airfoil), the flow domain with suitable dimensions and shape was designed and divided into six regions, as shown in Figure 1, to control the mesh. Through the Airfoil Tools website, the x and y coordinates of the NACA 0012 airfoil’s profile were obtained.

Using the Microsoft Excel and Fluent programs, the NACA0012 airfoil geometry was designed inside the domain with dimensions of 168 mm the chord length and 20.16 mm the airfoil’s maximum thickness at 30% of the chord from the leading edge. Then, the domain regions were named as the inlet, airfoil and outlet regions. This process was repeated for each modification case when adding the triangular rib to the airfoil’s surface.
2.2 Rib Geometry

The triangular rib geometry with size of 2%c (h=w=2%c=3.36mm) was designed on the upper surface of the airfoil at different locations, i.e. 50%c, 70%c and 90%c from the leading edge (one location for each modification case). The c character is the airfoil chord length.

Figure 2 explains the triangular rib geometry on the airfoil surface at 90%c from the leading edge, where h and w are the height and width of the triangular rib, respectively.

2.3 Mesh Generation

It is known that accuracy and convergence of the results depends the mesh (number of cells) and as the mesh is smoother as it is obtained on more accurate and convergent results but, in the case of very smooth mesh (very large cell number) the solution will take a very long time to achieve. Thus, the flow domain was divided into several regions to reduce the total mesh, the airfoil region was refined greatly and the regions away from the airfoil were slightly rough but the mesh smoothness increased towards the airfoil region gradually, as shown in Figure 3. It was obtained on a Wall Y plus value less than 0.75, as shown in Figure 4.

The Wall Y plus is a dimensionless parameter that indicates accuracy and convergence of the results and its value must be less than one to minimize errors: "the most desirable value is less
than one or the wall $Y+$ must be small enough to capture the flow characteristics " [12]. As its value reduces, as the vertical distance between the airfoil surface and first node ($Y$) reduces therefore, the errors will reduce. By the following equations, the distance ($Y$) can be calculated [12].

$$ Y = \frac{Y^+\mu}{\rho u_s} \quad \cdots (1) $$

Where:

$$ u_s = \sqrt{\frac{\tau_w}{\rho}} \quad \cdots (2) $$

$$ \tau_w = C_f \cdot 0.5 \rho u_{\infty}^2 \quad \cdots (3) $$

$$ Re_x = \frac{\rho u_{\infty} x}{\mu} \quad \cdots (4) $$

$$ C_f = [2 \log Re_x - 0.65]^{-2.3} \text{ for } Re_x < 10^9 \quad \cdots (5) $$

In this research $u_{\infty} = 7 \, m/s$, $x = 0.168 \, m$, $\rho = 1.21 \, kg/m^3$, $\mu = 1.8243 \times 10^{-5} \, kg/(m \cdot s)$ and $Re_x = 78000$ then, $Y = 2.66 \times 10^{-5} m$ which indicates that the first node is away from the airfoil surface by a distance equalling 0.0266 mm.
2.4 Governing Equations

The governing equations of the flow around the airfoil surface, according to the assumptions mentioned in paragraph 2 and the turbulence model (k-ω SST), are the following: [5]

i. Continuity equation:
\[
\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} = 0
\] … (6)

ii. Momentum (Naiver-Stokes) equations:
\[
\rho u \frac{\partial u}{\partial x} + \rho v \frac{\partial u}{\partial y} = - \frac{\partial p}{\partial x} + \frac{\partial}{\partial x} \left[ \mu \left( \frac{\partial v}{\partial x} + \frac{\partial u}{\partial y} \right) \right]
\] … (7)
\[
\rho u \frac{\partial v}{\partial x} + \rho v \frac{\partial v}{\partial y} = - \frac{\partial p}{\partial y} + \frac{\partial}{\partial x} \left[ \mu \left( \frac{\partial v}{\partial x} + \frac{\partial u}{\partial y} \right) \right]
\] … (8)

iii. Transport equations for k-ω SST model:
\[
\frac{\partial}{\partial t} (\rho k) + \frac{\partial}{\partial x_i} (\rho u_i k) = \dot{F}_k - \beta^* \rho k \omega + \frac{\partial}{\partial x_i} \left[ \mu + \sigma \mu_e \frac{\partial k}{\partial x_i} \right]
\] … (9)
\[
\frac{\partial}{\partial t} (\rho \omega) + \frac{\partial}{\partial x_i} (\rho u_i \omega) = \alpha \rho S^2 - \beta \rho \omega^2 + \frac{\partial}{\partial x_i} \left[ \mu + \sigma \mu_e \frac{\partial \omega}{\partial x_i} \right] + 2(1 - F_k) \rho \sigma \omega \frac{1}{\omega} \frac{\partial k}{\partial x_i}
\] … (10)

Where, S is the strain rate invariant measure, F1 is the function of blending, β* is 0.09 and σω2 is 0.856. Pk is production limiter used in the transport turbulence model of Menter’s shear stress to avoid the build-up of turbulence in areas of stagnation [5]. In addition, all of the constants are calculated by a blend from the consistent constant of the (k-ω SST) model by σ k, σ ω & etc. [5].

2.5 Setup and Solution

In the setup and solution, the turbulence model and the solution method are selected, respectively. In this research the k-ω SST (shear stress turbulence) model was selected as turbulence model since it is a model widely used for research of in field of aerodynamics. For the solution method, SIMPLE method was selected since it is an active method and gives
convergence and accuracy in the results, in less time and iterations, than other methods. Figure 5 illustrates.

In the setup, the initial and boundary conditions were entered into the Fluent program. In this research, all tests of the flow around the NACA0012 airfoil were carried out under the following conditions:

a. Constant air velocity at input of the domain \( u_\infty = 7 \text{ m/s} \),
b. Constant air density \( \rho = 1.21 \text{ kg/m}^3 \),
c. Constant air dynamic viscosity \( \mu = 1.8243 \times 10^{-5} \text{ kg/m s} \),
d. Constant Reynolds number \( \text{Re} = 78,000 \),
e. Constant air temperature \( \text{T} = 298 \text{ k} \) and
f. Turbulence intensity \( \text{I.T.} = 0.1 \),

![Figure (5): The lift and drag coefficients against the iterations.](image)

2.6 Mesh-Independence Test

Multi tests of the mesh were executed at angle of attack of 10-degree and freestream velocity of 7 m/s \( \text{Re} = 78,000 \) for choosing a suitable number of cells to obtain on more accurate and stable results of the lift and drag coefficients where, it was concluded that the mesh with the
number of 400,000 is the optimum for investigating the flow around NACA 0012 airfoil as shown in Table 1 and Figure 6.

Table 1: Effects of the mesh on the lift and drag coefficients.

| Solution time | Iterations | The mesh | Y+  | C_L    | C_D    |
|---------------|------------|----------|-----|--------|--------|
| 1 hour        | 1200       | 100000   | 3.5 | 0.585664 | 0.088656 |
| 2 hours       | 2500       | 200000   | 2.2 | 0.646658 | 0.083323 |
| 3 hours       | 3700       | 300000   | 1.4 | 0.667588 | 0.078056 |
| 4 hours       | 4900       | 400000   | 0.7 | 0.688000 | 0.072000 |
| 5 hours       | 6200       | 500000   | 0.6 | 0.682018 | 0.071815 |

Figure 6: The lift and drag coefficients against the mesh number.

2.7 Validation
After selection of the turbulence model, the solution method and the initial and boundary conditions and testing of the mesh independence, the lift and drag coefficients were calculated and the lift coefficient results of the present work and previous studies[1 and 8] were compared for a wide range of the attack angles. Figure 7 and it should be noted that that the research in [1] is a numerical study and [8] is an experimental study.

We noted a deviation of the present work’s results in relation to those from ref. [8], especially at the high angles of attack. Notably, the Reynolds number is the same for the two works (78,000) due to the test section blockage and the wing presence inside it, and other experimental errors for ref. [8]. Meanwhile, results from ref. [1] results are convergent with the present work’s, although the Reynolds number differs slightly for the two (78,000 for the present work and 100,000 for ref. [1]). This suggests that the present work’s results are accurate and correct. Further, it was noted that, at the low attack angles (α<14-degree), the present work’s results are intermediate between those of refs. [1] and [8] as shown in Figure 7. Where, α is the attack angle (degree).
3. Results and Discussion

All the air flow tests around the NACA0012 airfoil surface were done at constant Reynolds number (Re=78000) which corresponds to 7 m/s freestream velocity and different attack angles (0˚, 10˚, 12˚, 14˚, 16˚, 18˚, 20˚ and 22˚) to calculate the lift and drag coefficients without and with the triangular rib at different locations of 50%c, 70%c and 90%c from the leading edge of the airfoil, with size of 2%c. Through results of the lift and drag, the airfoil’s performance was calculated by dividing the lift on the drag.

The results showed, for the smooth airfoil, that as the attack angle increases as the lift and the drag coefficients increase until the angle of 14˚, at which the airfoil produces maximum lift but, after that angle the lift falls and drag continues in the rise as shown in Figures 8a and 8b) and this is known by the stall condition, which occurs due to the air flow separation from the airfoil’s upper surface entirely, and occurrence of strong vortices causing adverse flow of the air stream.

Figure 8c shows results of the airfoil’s aerodynamic performance (the lift to drag ratio) against the angle of attack, where it is noted the performance rise with increase of the attack angle until the angle of 10˚, at which maximum value of that performance is obtained and after that angle it reduces because of increase of the drag dramatically.

While, for the modified airfoil with the triangular rib, the results indicated that:

At low angles (less than 12˚), the rib’s presence at all locations has negative effect where it causes increase in the drag and reduction in the lift and therefore, large reduction in the airfoil performance specially the location of 50%c. At these angles, there is no separation of the air flow, therefore the rib will cause drag of the flow and generate vortex of the downstream flow causing an adverse pressure gradient.

At the angle of 12˚, the rib effect, for the lift results is positive only at the location of 90%c whereas, for the drag results, at all the locations is negative thus, only the location of 90%c can
improve the airfoil’s performance at that angle since no separation of the airflow at the two locations of 50%c and 70%c.

At the higher angles \(12^\circ<\alpha<22^\circ\), all results of the lift, drag and performance of the airfoil, for all the locations, are positive. Using the rib gives two advantages, which are significant increase in the lift coefficient and reduction in the drag coefficient therefore, there is an important enhancement in the airfoil’s performance as shown in Figures 8a, b and c, since, at these angles, the flow separation bubble becomes significant and the rib presence will break it causing recirculation of the adverse flow and reattachment of the separated upstream flow.

But at the attack angle of 22°, the triangular rib’s presence on the airfoil’s surface has a very small effect for all the results concerning the lift, drag and performance of the airfoil. This is shown in Figures 8a, b and c because the separation vortex size becomes very large compared with the rib size used.
Figure 8a: The lift coefficients versus the attack angle without and with the triangular rib at the locations of 50, 70 and 90% c with 2%c size.

Figure 8b: The drag coefficients versus the attack angle without and with a triangular rib at the locations of 50, 70 and 90% c with 2%c size.
Figure 8 c: The lift to drag ratios versus the attack angle without and with the triangular rib at the locations of 50, 70 and 90%c with 2%c size.

Figure 9 shows results of the pressure coefficients around the NACA0012 airfoil’s surface without and with the triangular rib at different locations of (50, 70 and 90) %c from the leading edge with 2%c size at 14˚ angle of attack.

It was noted that, for the three modification cases, the pressure on the upper surface dropped with the rib’s presence and the flow separation area reduced, since it broke the adverse flow vortices causing reattachment of the separated airflow and as the rib location approached the trailing edge as the flow reattachment area increased.

Figure 9: The pressure coefficients around the airfoil’s surface without and with the triangular rib at the locations of 50, 70 and 90 %c with 2%c size at angle of 14˚.

4. Conclusions

Comparison of the results of the flow investigation of the modified NACA0012 airfoil’s surface with the triangular rib for the three locations (50%c, 70%c and 90%c) and the size of 2%c,
with results of the smooth case, it was concluded that use of that technique largely depends on the angle of attack. Using it at low angles (less than 12-degree), leads to negative results (lift reduction and drag increase and therefore, decay in the airfoil’s performance). At the angles above 12 degrees, the rib’s effect, at all three locations, becomes useful for the lift, drag and performance of the airfoil while. At 12 degrees, it is useful only for the location of 90%c where the lift increases and therefore, performance improves.

Also, it was concluded that selection of the rib location greatly depends on the angle of attack where, at the 14-degree angle, the modified airfoil with the rib of the location of 90%c produced a higher lift and performance and less drag in comparison with the smooth case and other cases. Whereas, for the higher angles, using the rib at the location of 50%c produces a higher lift and enhances the performance and delays the stall angle two degrees (from 14˚ to 16˚).

References

[1] L. Huang, P. G. Huang, and R. P. LeBeau, 2003, “Numerical study of blowing and suction control mechanism on NACA 0012 airfoil” Journal of Aircraft Vol. 41, No. 5, pp 1005–1013.

[2] Hua Shan, Li Jiang, Chaoqun Liu, Michael Love, and Brant Maines, 2007, “Numerical study of passive and active flow separation control over a NACA0012 airfoil” Computers & Fluids 37 (2008) 975–992.

[3] Y.F. Lin, K.Lam, L.Zou b and Y.Liu, 2012, Elsevier, “Numerical study of flows past airfoils with wavy surfaces” Journal of Fluids and Structures Vol. 36 136–148.

[4] Deepanshu Srivastav, BITS (2012), “Flow control over airfoils using different shaped dimples”, International Conference on Fluid Dynamics and Thermodynamics Technologies (FDIT 2012) IPCSIT vol.33, IACSIT Press, Singapore.
[5] Kianoosh Yousefi, S. Reza Saleh and Peyman Zahedi, 2013, “Numerical study of flow separation control by tangential and perpendicular blowing on the NACA 0012 airfoil” *International Journal of Engineering*, Vol. 7), Issue 1.

[6] Syed Hasib Akhter Faruqui, Md.Abdullah AlBari, MdEmran and Ahsan Ferdaus, 2014, “Numerical analysis of role of bumpy surface to control the flow separation of an aerofoil.” Tenth International Conference on Mechanical Engineering, ICME 2013, *Procedia Engineering* 90,255–260.

[7] Pritesh S. Gugliya1, Yogesh R. Jaiswal, Akshay B. Chhajed, Sachin V. Jain and Hitesh R. Thakare, 2015, “CFD analysis of airfoil NACA0012” *International Journal of Modern Trends in Engineering and Research*.

[8] Kanok Tongsawang, 2015, “Stall Control of a NACA0015 aerofoil at low Reynolds numbers”, Thesis, University of Sheffield.

[9] Ahmed S. Shehataa, Qing Xiaoa, Khalid M. Saqrc, Ahmed Naguibb and Day Alexandra, 2017, “Passive flow control for aerodynamic performance enhancement of airfoil with its application in Wells turbine – under oscillating flow condition”, *Ocean Engineering* 136 pp.31–53.

[10] Sandesh K. Rasal and Rohan R. Katwate, 2017, “Numerical Analysis of lift & drag performance of NACA0012 wind turbine aerofoil” *International Research Journal of Engineering and Technology* Vol. 04, Issue 06.

[11] Amit Kumar Saraf, Mahendra Pratap Singh, Tej Singh Chouhan, 2017, “Effect of dimple on aerodynamic behaviour of airfoil” *International Journal of Engineering and Technology* Vol. 9 Issue 3 pp. 2268–2277.