Dijlah University college
Department of cooling and air-conditioning techniques engineering
1 abbas.aldoory@duc.edu.iq
2 haider.lafta@duc.edu.iq

Improving Aerodynamic Coefficients of an airfoil using suspended wing

Abbas Fadhil Mahmood[1] & Haider Lafta Aneed[2]

Abstract:

In this paper, the aerodynamic force coefficients of the airfoil were evaluated numerically. The airfoil is geometrically identical to the NACA 64A012 airfoil. The study was done on this airfoil with and without suspended wing above the leading edge. It was found numerically that both lift and drag coefficients increase by adding of suspended wing. The pressure and velocity distribution was obtained in order to locate the separation point in addition to the overall aerodynamic characteristics. The CFD code (FLUENT 6.2) packages were utilized for simulating the flow around the airfoil. The control of the plane was done by the end of the airfoil (flaps), if an early separation of the flow occurred, will be a loss of control over the forces of lifting and braking of the aircraft, therefore a wing stabilizer added to delay the separation point and improve the aerodynamic characteristics of the airfoil.

The results illustrate the benefits of adding suspended airfoil above the leading edge of the airfoil, the separation was removed from the trailing edge at the same angle of attack in both cases. The lift coefficient increased by about (32%) at the maximum angle of attack, while the drag coefficient increased by about (20%) only.

Keywords. Suspended wing, Separation Point, Lift Coefficient, Drag Coefficient

Introduction:

The continuous increase in the control of the aircrafts necessitates continuous improvement in the performance of aerodynamic coefficients of the airfoil. All solid objects traveling through a fluid (or alternatively a stationary object exposed to a moving fluid) acquires a boundary layer of fluid that so rounds them where viscous forces exist in the layer of fluid adjacent to the solid surface. Boundary layers may be either laminar or turbulent. In order to determine whether the boundary layer is laminar or turbulent, the Reynolds number of the local flow conditions is calculated.

When the boundary layer travels against an adverse pressure gradient, the pressure increases and the velocity decrease in the flow direction. When the velocity becomes zero, flow separation occurs.. [1] [2] The fluid flow becomes detached from the surface of the object, and eddies and vortices are formed. In aerodynamics, flow separation can often lead to an increase in pressure drag, which is caused by the pressure differential between the front and rear surfaces of the object as it travels through the fluid. For this reason much effort and research has gone into the design of aerodynamic and hydrodynamic surfaces which delay flow separation and keep the local flow attached for as long as possible. Roughening of golf balls, stitching baseballs and vortex generators are examples of boundary layer control structures for controlling the separation pattern.

When the flow direction of the boundary layer portion that is adjacent to the wall or leading edge is reversed, the separation occurs. The point between the forward flow and backward flow is called the separation point. The shear stress at this point is zero. The overall boundary layer thickness increases suddenly at the separation point and the reversed flow at its bottom the reversed flow of its bottom forces it off the surface [4].

Adverse pressure gradient:

The outer flow potential imposes adverse pressure gradient which in turn causes flow the outer potential flow. The equation of momentum inside the boundary layer in the flow direction can be written as,

$$u\frac{\partial u}{\partial x} = -\frac{1}{\rho}\frac{dp}{dx} + \nu\frac{\partial^2 u}{\partial y^2} \qquad (1)$$

Where *x, y* are the coordinates in the flow and normal directions respectively. The velocity decreases when the adverse pressure gradient $\frac{\mathrm{dp}}{\mathrm{dx}} > 0$. If the adverse pressure gradient is strong enough, the velocity may go to zero [5].

Influencing parameters:

The possibility of a boundary layer separation primarily depends on the of the adverse or negative edge velocity gradient distribution, $du_o/dx\ (x) < 0$ Along the surface, which is proportional to the pressure and its gradient and is governed by the differential form of the Bernoulli equation, which represents the equation of momentum for the outer inviscid flow,

$$\rho u_o \frac{du_o}{dx} = -\frac{dp}{dx} \qquad (2)$$

But the adverse magnitudes of $\frac{du_o}{dx}$ required for separation for turbulent flow are much greater than that for laminar flow,. Reynolds number also affects the separation resistance. In turbulent flow, the separation, resistance increases as the Reynolds number increases for a given distribution of adverse $\frac{du_o}{dx}$. Reynolds number has no effect on the boundary layer separation, resistance for laminar flow.

Effects of boundary layer separation,

When the separation occurs, the displacement thickness of the boundary layer increases sharply. Both the pressure field and outside potential flow are modified due to that increase. In the case of airfoils, the modification in the pressure field leads to pressure increase, which is undesirable. Sharp modification can also lead to loss of lift and stall, which are also undesirable. In case of internal flows, undesirable phenomena such as compressor surge and flow losses increase occur due to flow separation [6].

Shedding vortices are also effects of boundary layer separation. This is known as Ka'rma'n Vortex Street. Vortices start shedding off at a certain frequency which could cause vibrations in the structure which they shed off. When the shedding vortices shed off at the structure resonance frequency, serious structural failures may occur.

Computational fluid dynamics (CFD).

CFD, is a branch of fluid mechanics in which numerical methods and algorithms are used in solving and analyzing problems which involve fluid flow. Calculations required for the simulation and interaction of liquids and gases with surfaces that are defined by boundary conditions are performed using computers. Researches that aim at software improvement in terms of the accuracy and speed of complex simulations such as transonic or turbulent flows. This software is Initially validated experimentally using a wind tunnel. Final validation coming in full-scale testing is then performed, e.g. flight tests.

Assumptions.

1. Air is at standard conditions.
   a. Pressure = 1 atm .
   b. Temperature = 25 °C .
   c. Density = 1.225 kg/m^3.
2. The flow around the airfoil is turbulent.
3. There are no slip conditions at the surface of airfoil and wing stabilizer.

Governing equations of fluid flow.

The fluid flow is governed by the conservation of mass (Continuity equation) which states that the fluid mass is conserved and conservation of momentum (Navier-Stokes equations) which states that the change in the fluid momentum is equal to the sum of forces..

Continuity

$$\frac{\partial(\rho u)}{\partial x}+\frac{\partial(\rho v)}{\partial y}+\frac{\partial(\rho w)}{\partial z}=0 \qquad (3)$$

Momentum

In – X direction

$$\frac{\partial(\rho uu)}{\partial x}+\frac{\partial(\rho uv)}{\partial y}+\frac{\partial(\rho uw)}{\partial z}=\frac{\partial}{\partial x}\left[\mu_{eff}\frac{\partial u}{\partial x}\right]+\frac{\partial}{\partial y}\left[\mu_{eff}\frac{\partial u}{\partial y}\right]+\frac{\partial}{\partial z}\left[\mu_{eff}\frac{\partial u}{\partial z}\right]+\rho g_x-\frac{\partial p}{\partial x}$$

(4)

In – Y direction

$$\frac{\partial(\rho vu)}{\partial x}+\frac{\partial(\rho vv)}{\partial y}+\frac{\partial(\rho vw)}{\partial z}=\frac{\partial}{\partial x}\left[\mu_{eff}\frac{\partial v}{\partial x}\right]+\frac{\partial}{\partial y}\left[\mu_{eff}\frac{\partial v}{\partial y}\right]+\frac{\partial}{\partial z}\left[\mu_{eff}\frac{\partial v}{\partial z}\right]+\rho g_y-\frac{\partial p}{\partial y}$$

(5)

In – Z direction

$$\frac{\partial(\rho wu)}{\partial x}+\frac{\partial(\rho wv)}{\partial y}+\frac{\partial(\rho ww)}{\partial z}=\frac{\partial}{\partial x}\left[\mu_{eff}\frac{\partial w}{\partial x}\right]+\frac{\partial}{\partial y}\left[\mu_{eff}\frac{\partial w}{\partial y}\right]+\frac{\partial}{\partial z}\left[\mu_{eff}\frac{\partial w}{\partial z}\right]+\rho g_z-\frac{\partial p}{\partial z}$$

(6)

Where μ_{eff} Is the effective viscosity which combines laminar and turbulent stresses as?

$\mu_{eff}=\mu+\mu_t, \mu_t=\frac{C\mu\rho k^2}{\varepsilon}$.

Turbulence modeling theory:

Turbulence from the standpoint of modern aero hydromechanics represents a fully developed area of knowledge, containing useful information for engineering practice. Nowadays, there is a widespread technological complex program. Despite significant progress in model development, theoretical turbulence designs as basic science still far from its completion. So, in 1998. In Oxford, at an international seminar on problems of computational hydrodynamics simulation of turbulence (namely, direct numerical modeling and modeling by big vortexes) it was recognized one of the three actual scientific directions (together with the decision conjugate tasks aeromechanics and environmental problems). It should be noted that during the past half-century, many aero hydromechanics scientists have contributed into evolution views on turbulence and today during the industrial development, computational hydrodynamic tens and hundreds of thousands of specialists around the world are engaged in the calculations of various turbulent flows..

The standard k-ε model.

The standard k-ε model was used in this work. It is one of the most popular and most widely used turbulence models. This model is semi-empirical. It consists of two transport equations. One is for specific turbulent kinetic energy (k) and the other is for the turbulent dissipation rate (ε).

Transport equations for standard k-ε model.

For turbulent kinetic energy (k),

$$\frac{\partial}{\partial t}(\rho k)+\frac{\partial}{\partial x_i}(\rho k u_i)=\frac{\partial}{\partial x_j}\left[\left(\mu+\frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right]+p_k+p_b-\rho\varepsilon-Y_M+S_k \quad (7)$$

For dissipation ε

$$\frac{\partial}{\partial t}(\rho\varepsilon)+\frac{\partial}{\partial x_i}(\rho\varepsilon u_i)=\frac{\partial}{\partial x_j}\left[\left(\mu+\frac{\mu_t}{\sigma_\varepsilon}\right)\frac{\partial\varepsilon}{\partial x_j}\right]+C_{1\varepsilon}\frac{\varepsilon}{k}(p_k+C_{3\varepsilon}p_b)-C_{2\varepsilon}\rho\frac{\varepsilon^2}{k}+S_\varepsilon$$

(8)

Modeling turbulent viscosity

Turbulent viscosity is modelled as,

$$\mu_t = \rho C_\mu \frac{k^2}{\varepsilon}$$

Production of k

$$p_k = -\rho\overline{u_i' u_j'}\frac{\partial u_j}{\partial x_i}$$

$$p_k = \mu_t S^2$$

Where *S* is the modulus of the mean rate of strain tensor, defined as:

$$S \equiv \sqrt{2S_{ij}S_{ij}}$$

Model constants

$C_{1\varepsilon} = 1.44, C_{2\varepsilon} = 1.92, C_{3\varepsilon} = -0.33,\ C_\mu = 0.09, \sigma_k = 1, \sigma_\varepsilon = 1.3$ [9]

Computer program.

The software used in this work was the FLUENT(6.2) code and GAMBIT code.

FLUENT code

One of the commercial program for solving fluid and heat transfer problems is FLUENT code, for computation of fluid dynamics (CFD).

FLUENT is an ultimate useful computer based program that is used to model fluid flow and heat transfer problems in complex geometries. Using this program, a complete mesh flexibility for solving such problems with unstructured meshes which about complex geometries may be simply generated . Two processors are used to solve the flow. The first one is the program structure by which the geometry and grid are created using GAMBIT. The second one solves continuity, momentum, and energy equations. While EULER equations drop the viscosity term by using the finite volume technique, and by using the initialization to describe with some details about the implementation of the boundary condition. Two methods are used in the solver: explicitly or implicitly. Explicit methods are generally easy to code. However, very small time steps are required to meet stability requirements. This increases the number of iterations and computation time required to reach the steady state solution. The implicit scheme simultaneous solves several equations. In this scheme, The maximum time step allowed is much larger than that for the explicit scheme which reduces the number of iterations and computation time required [10]

Results and Discussion.

The purpose of these simulations is to study the effects of adding the wing stabilizer above the leading edge of the airfoil. For the Maneuvering, take off, and landing configuration, usually a larger maximum lift coefficient pursued. Figure (1) Shows that the lift coefficient increased by 32% at the maximum angle of attack when we added the small wing above the leading edge. Figure (2) Shows that the drag coefficient increased by 20% only. Figure (3&4) shows the relationship between (drag - lift coefficient) and the angle of attack, it obviously increases in wing efficiency ($\eta = \frac{CL}{CD}$) by14%.

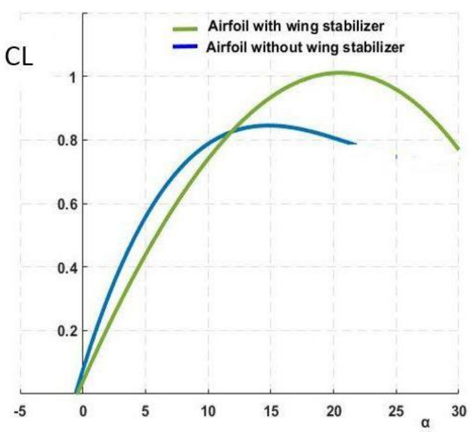


Figure (1) lift coefficient with angle of attack

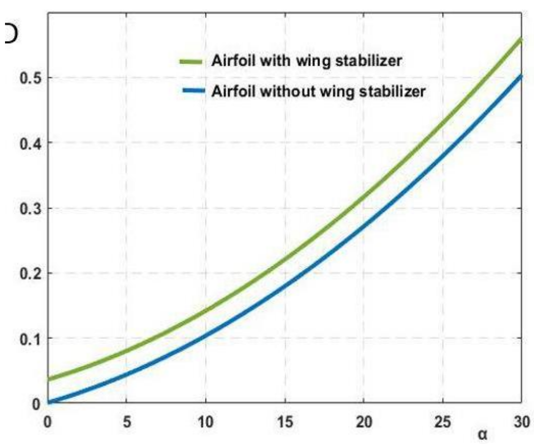


Figure (2) drag coefficient with angle of attack

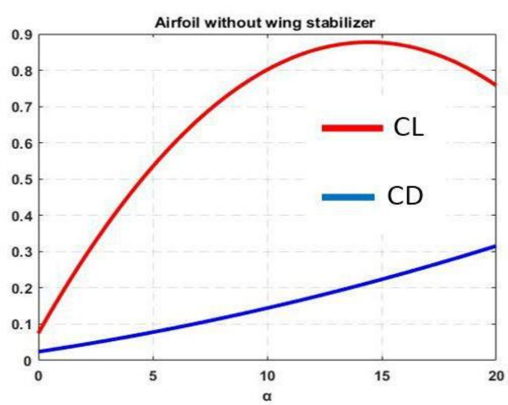
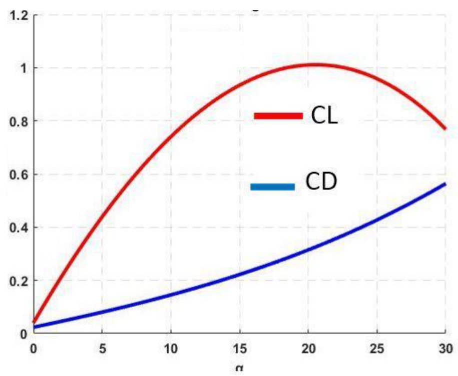
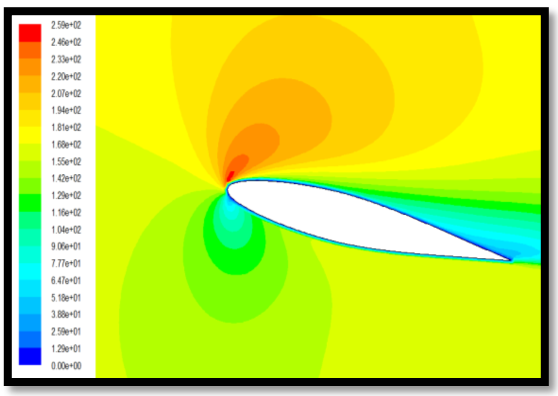


Figure (3) Lift and drag coefficients without suspended wing Figure

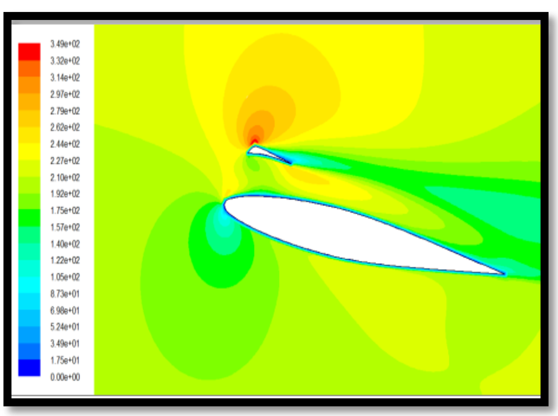


(4) Lift and drag coefficients with suspended wing

As shown in figures (5, 6, 7) velocity distribution of two cases simulation, with and without suspended wing, the separation point delayed and vary with angle of attack (10, 15, 20) respectively using suspended wing

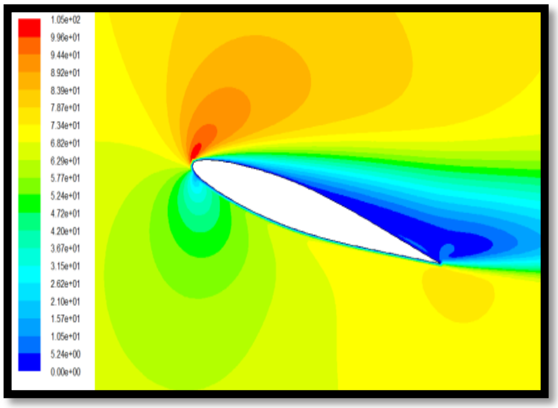


(a)



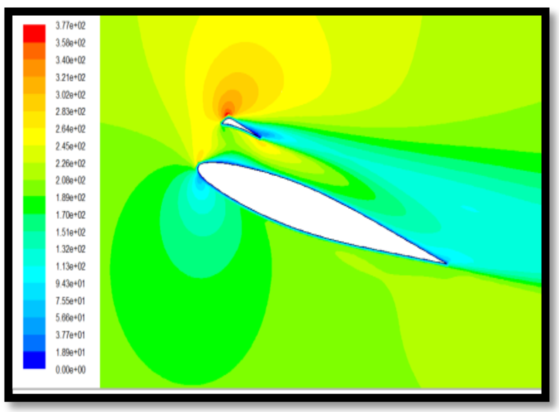
(b)

Figure (5) Velocity distribution at 10 degree(a) without suspended wing (b) With suspended



wing.

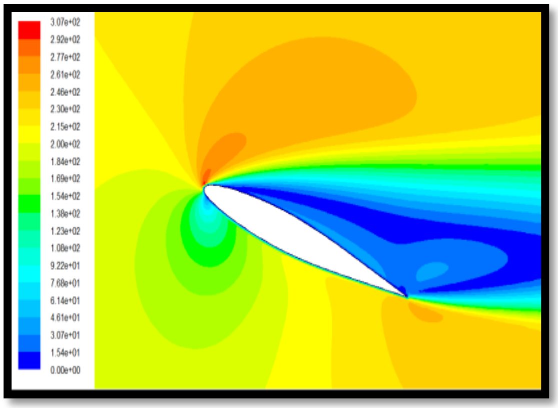
(a)



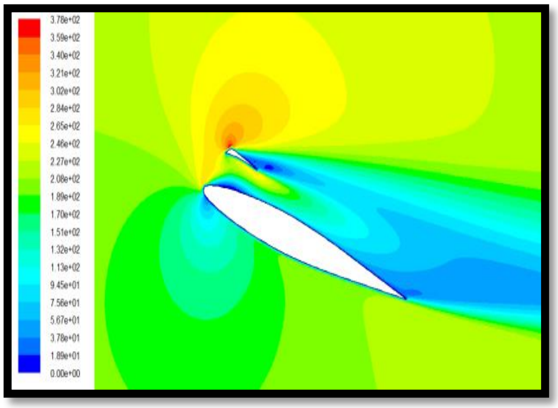
(b)

Figure (6) Velocity distribution at 15 degree

(a) without suspended wing (b)
With suspended wing.



(a)



(b)

Figure (7) Velocity distribution at 20 degree a) without suspended wing (b) With suspended wing.

References:

[1] J. Driver, D. W. Zingg – "Optimised natural laminar flow Airfoils", AIAA paper 2006-247, Jan. 2006

[1] J. Anderson, "Introduction to Flight 7th edition", McGraw-Hill Science/ Engineering/ Math, (2011), Section 4.20.

[2] Clancy, L.J., Aerodynamics, Section 4.14.

[3] White (2010), "Fluid Mechanics", Section 7.1 (7th edition).

[4] Wilcox, David C. Basic Fluid Mechanics. 3rd ed. Mill Valley: DCW Industries, Inc., 2007. 664-668.

[5] Balmer, David. "Separation of Boundary Layers." School of Engineering and Electronics. 2 December 2007. University of Edinburgh. 12 March 2008 <http://www.see.ed.ac.uk/~johnc/teaching/fluidmechanics4/2003-

04/fluids14/separation.html>.
[6] Fielding, Suzanne. "Laminar Boundary Layer Separation." 27 October 2005. The University of Manchester. 12 March 2008 <http://www.maths.manchester.ac.uk/~suzanne/teaching/BLT/sec4c.pdf>.
[7] Launder B.E. and Spalding D.B.,"the numerical computational of turbulent flow " computer methods in applied mechanics engineering (3) pp. 269-289.
[8] J. D. Hoffman, Numerical Methods for Engineers and Scientists, Marcel Dekker, 2001 (QA297 .H588).
[9] http: / /www.cf d- on line.com /Wiki / Standard_k- epsilon _model.
[10] Fluent 6 Users Guide, fluent Inc., (2001).