Numerical simulation of a new spoiler on upper surface of Clark Y14 wing

Mohammed Kheiraldeen Abbas Askar¹, Amer Q Hameed², Kadhim H Suffer³, Zuradzman M Razlan⁴

¹,³ Mechanical Engineering Department, Al-Nahrain University, P.O. Box 64040, Baghdad, Iraq
²Mechanical Engineering Department, Dyaala University, Dyalla, Iraq
⁴School of Mechatronic Engineering Universiti Malaysia Perlis, Pauh Putra Campus, 02600 Arau, Perlis, Malaysia

¹dr.mohammedaskar@yahoo.com, ²amerkader93@gmail.com, ³kadhim_askar@yahoo.com, ⁴zuradzman@unimap.edu.my

Abstract. Spoilers are one type of devices that used to reduce the lift on the aircraft wing. A theoretical and numerical study involves simulation of the flow behaviour around of airfoil type (Clark y-14) for the wing section are carried out. ANSYS14 software is used to solve the governing equation in three dimensions steady incompressible turbulent regime with appropriate turbulent model (k – ω, SST). In this study the lift and drag for different angles of attack, with and without spoiler effect on distribution of pressure and velocity distribution on the wing surface are steadied. A constant velocity 40 m/s is used on the wing with and without new spoiler have different cord (10% C) and (15% C) and angles of attack from -8° to +18º. The wing model is rectangular with (30 cm span and 9 cm cord). Wing has aspect ratio 30/9 and a maximum thickness of 14 % of the wing chord. A trial and error for five-gaps between spoiler and wing (0%, 2.77%, 3.33%, 3.88% and 6.66% of the wing cord) are carried out when is the wing cord is equal to 9 cm and for different gaps (0, 2.5, 3, 3.5 and 6 mm). Theoretical results indicated when is location spoiler 50%C, δ=0, new spoiler cord (HI) 10%C and gap between new spoiler and wing equal 3.33% C which increase maximum lift theoretically to (28.29%) compared with clean wing.

Keywords: Rectangular wing, new spoiler, ANSYS software, lift, drag, high drag device.

1. Introduction

Spoiler is the one type of high drag devices that used to reduce the lift on the aircraft wing [1]. It is like flap structure of rectangular cross-section whose leading edge is hinged on upper wings (at any angle) disturbing the streamlined flow of air there by changing the amount of lift. The net force on aircraft in the vertical direction is equal to weight minus lift. Thus, when the spoilers are employed lift gets reduced as a result the net downward force acting on the wing increases. There are two types of spoiler used in aircraft, Flight Spoilers and Ground Spoilers [2]. The flight spoilers are used to reduce lift and increase drag, without increasing speed to a very high rate this allows easier landing the aircraft, while ground spoilers also called lift dumpers (or called airbrake) deployed on landing of the aircraft. These types are slow down the aircraft by increasing the drag and also decrease the lift gradually. As a result, all the aircraft weight is moved from the wings to the undercarriage. This leads to ease of braking also reduces chances of skidding. To get a clear understanding of the work of the spoiler, and its relationship to lift and drag, and the development of mechanisms spoiler for the best result of the corruption of many of the research conducted in the past to investigate this matter. Some of studies are focused on the
experimental work for the spoilers [3-6], while the other studies was carried out by use numerical simulations [7-13]. From previous studies, it can be seen that there are many types of investigations and studies for the effect of plain spoiler at different positions and different heights and some configurations of spoiler on the aerodynamic characteristics of different wing sections experimentally and numerically by use CFD software. There is no study include the gap between the spoiler and wing.

2. Current simulation
The aim of this study is to analysis the airflow around the wing with and without new spoiler to control the separation of the flow over the wing at high angle of attack. The new spoiler in this steady means to change the position (gap) and the inclination angle of spoiler with respect to wing for control the separation. Because of the complication of airfoil surfaces with spoiler configurations and the strong viscous fluid effects it is difficult to obtain an analytical solution of the Navier- stokes equation for such complex configurations. Thus the numerical techniques have been used to solve this equation. The analysis of the flow field around airfoil with and without new spoiler will be carried out by using a numerical techniques using a computational fluid dynamics (CFD).

AutoCAD software is used to modelling the 3-D wing by using the Clark y-14 airfoil data for 2-D and then extrude the profile of this data 30cm to get 3-D wing and then draw spoiler over this wing as shown in Figure 1. The computational domain of the test section is fixed as (30.48cm x 30.6 cm x 60.96cm) as shown in Figure 2. In the previous studies the wing model with and without spoiler were tested at different angels of attack ranged from (-8° to 18°) [14]. In this study the wing model used was a rectangular wing of Clark y-14 airfoil, and had a maximum thickness (14 % C) and an aspect ratio (δ) of (30/9), the wing model used has 30cm span and 9cm chord. The model is meshed as a fine mesh with min. size 0.0000002m and max. Size 0.000002m for the wing with and without new spoiler as shown in Figure 3 (a) and 3 (b) respectively.

![Figure 1. 3-D wing model by using the Clark y-14 data with spoilers.](image1)

![Figure 2. The domain of the test section.](image2)
3. Results and discussion

Fluid flow (fluent) software is used to solve the current study problem. In this analysis the governing partial differential equation in three dimensions, which based on conservation of momentum and conservation of mass equations will be solved. The (k-ω) turbulence model also will be used in this analysis to demonstrate the turbulence effect on the flow, because of its give a good behaviour in adverse pressure gradients and separating flow. The working fluid is air at temperature 17°C and the flow characteristics assumed to be as follows 3-D, incompressible flow, Newtonian and physical properties of the air are constant. The inlet is choice as inlet velocity and fixed to 40 m/s corresponding to Reynolds number of 8.24119x10^5, and the outlet is a pressure outlet. No slip boundary condition is specified for the wing with and without winglet surfaces and is set as a wall. Figure 4 shows the scaled residuals of continuity, momentum, k and ω as the solution progresses to convergence history. In order to illustrate the effect of best gap in the improvement of aerodynamic performance, simulations were performed for wing without and with best gap for angles of attack (A.O.A.) from (0° to 10°) in this test to study for steadying all sides. The gaps between spoiler and wing increase the lift coefficient of Clark y-14 airfoil wing compared with clean wing (wing without spoiler). Figure 5 shows the results of lift against the angle of attack (A.O.A.) for clean wing and wing with gap 3mm, these results indicate the gaps worked like nozzle work in other word increase air.

Figures (6) and (7) show the contours of steam line for the flow of air over the clean wing and the wing with best spoiler gap (3mm), with aspect ratio (δ = 0) and A.O.A. = 0°, while Figures (8) and (9) for aspect ratio (δ = 0) and A.O.A. = 10°. The velocity magnitude has maximum value at the upper surface especially at the leading edge, and low value at the lower surface led to increase pressure, the streamline flow during the gap shows in Figure 8. In Figure 6 when clean wing used, it can see that the streamline driving away from the wing at trailing edge compare with the results of Figure 7, the flow
streamline approach more the upper surface of the wing that effect allow flow rate remain laminar and prevent the separation. Frome these results it can be concluded that the effect of best gap and that effect more clear when the A.O.A. = 10° as shown in Figures (8) and (9). When spoiler angle = 90° we see the streamline before spoiler move up and down spoiler and after spoiler turbulent flow and random as seen in Figure 18. Velocity in the gap (compressed boundary layer) led to push any separation flow on upper surface of the wing to back the wing and worked to reduce separation of stream line on upper surface Figure (6) and (7). The addition of gap contributes to accelerate the flow on the upper surface of the wing all these reasons cause increases the maximum lift, compare with clean wing clear shown in Figure 5. Maximum lift can be get from spoiler position 50% Wing Chord, Spoiler height or spoiler cord 9% Wing Chord, and gap 3.33%.

Figures (10) and (11) show the contours of total pressure in (Pa) for the flow of air over the clean wing and the wing with best spoiler gap (3mm), with aspect ratio (δ = 0) and A.O.A. = 0°, while Figures (12) and (13) for aspect ratio (δ = 0) and A.O.A. = 10°. From the results of Figure 11 it can be seen that the total and static pressures for wing with and without spoiler decrease at upper surface of spoiler and increase pressure lower surface of the wing. When the spoiler at the angle of 90°, the increases of the pressure on upper surface of the spoiler led to increase the drag as shown in Figure 19.

Figures (14) and (15) show the contours of the effect of viscosity in (kg/m.s) for the flow of air over the clean wing and the wing with best spoiler gap (3mm), with aspect ratio (δ = 0) and A.O.A. = 0°, while Figures (16) and (17) for aspect ratio (δ = 0) and A.O.A. = 10°. In Figures (17) and (17) it’s clearly can see the effect viscosity at trailing edge its small than when the best spoiler gap used, and very high after spoiler when spoiler angle 90° and very small at trailing edge as shows in Figure 20.

Figure 6. The contours of stream line on clean wing for δ=0 and A.O.A= 0

Figure 7. The contours of stream line on wing with new spoiler for δ=0 and A.O.A= 0

Figure 8. The contours of stream line on clean wing for δ=0 and A.O.A= 10

Figure 9. The contours of stream line on wing with new spoiler for δ=0 and A.O.A=10
**Figure 10.** The contours of the total pressure (Pa) on clean wing for $\delta=0$ and A.O.A= 0

**Figure 11.** The contours of the total pressure (Pa) on wing with new spoiler for $\delta=0$ and A.O.A= 0

**Figure 12.** The contours of the total pressure (Pa) on clean wing for $\delta=0$ and A.O.A= 10

**Figure 13.** The contours of the total pressure (Pa) on wing with new spoiler for $\delta=0$ and A.O.A= 10

**Figure 14.** The contours of the viscosity effect (kg/m.s) on clean wing for $\delta=0$ and A.O.A= 0

**Figure 15.** The contours of the viscosity effect (kg/m.s) on wing with new spoiler for $\delta=0$ and A.O.A= 0
4. Conclusion
In this investigation the numerical simulation is carried out for the clean wing and wing with the spoiler
the three dimensional domain to investigate the aerodynamics flow. ANSYS FLUENT14. software’s is
used for this investigation with k-ω turbulence model. The main conclusions from this simulation are;
• The gaps between spoiler and wing increase the lift coefficient of Clark y-14 airfoil wing compared
with clean wing (wing without spoiler). Maximum lift can be get from spoiler position 50% Wing
Chord, Spoiler height or spoiler cord 9% Wing Chord, and gap 3.33%.
• Velocity in the gap (compressed boundary layer) led to push any separation flow on upper surface
of the wing to back the wing and worked to reduce separation of stream line on upper surface and
the addition of gap contributes to accelerate the flow on the upper surface of the wing all these
reasons cause increases the maximum lift, compare with clean wing
• When the spoiler at the angle of 90°, the increases of the pressure on upper surface of the spoiler led
to increase the drag.
• The effect viscosity at trailing edge its small than when the best spoiler gap used, and it’s very high
after spoiler when spoiler angle 90° and very small at trailing edge.
• Results reveal we can use Spoiler high lift device when spoiler angle 0° and high drag device when
spoiler employed. From this study indicates to the first time in the history of aircrafts can be
employed gap between spoiler and wing (used spoiler high lift device δ = 0 and high drag device
when spoiler employed) to improve wing performance.
References

[1] Theodore A T 1975 *Introduction to the aerodynamics of flight* National Aeronautics and Space Administration (Washington).

[2] Nair S, Nair S and Nair Sy 2014 Aircraft Braking System *Indira Gandhi Delhi technical* 4

[3] Mashud M, Ferdous M and Omee S H 2012 Effect of spoiler position on aerodynamic characteristics of an airfoil *International Journal of Mechanical & Mechatronics Engineering* 12 1-6

[4] Harley C D 2010 Aerodynamic performance of low from factor spoilers *School of Mechanical, Aerospace and Civil Engineering, University of Manchester, PhD Theses*.

[5] Doggett R V 1989 Some effects of aerodynamic spoilers on wing flutter *National Aeronautics and Space Administration* NASA, pp 193-209

[6] Blair G and Karamcheti K 1984 An experimental Study of airfoil-spoiler aerodynamics *Stanford University Department of Aeronautics and Astronautics Stanford, California 94305*

[7] Patel M P and Sowle Z H 2007 Autonomous sensing and control of wing stall using a smart plasma slat *Journal of Aircraft* 44 516-527

[8] Wentz H W 1975 Effectiveness of spoilers on the GA (W)-1 Airfoil with a high performance fowler flap *NASA Wichita state university Kansas 67208-1975*.

[9] C. S Barnes, Ph.D 1966 A developed theory of spoilers on airfoiles *Aeronautical research council current papers London C.P.NO.887*

[10] Sherman A, Clevenson and John E 1957 Experimental investigation of the oscillating forces and moments on a two-dimensional wing equipped with an oscillating circular-arc spoiler *American society of civil engineers ISSN (print): 0893-1321-1957*.

[11] Kondwani Kanjere .et al. Aeroacoustic 2010 Investigation of Deployed Spoiler during Steep Approach Landing, University of Southampton, UK.- AIAA 2010-3992

[12] Guillaume F, Pape M C, and Montagnac M 2004 Numerical simulation around wing control Surfaces 24*International congress of the aeronautical Sciences* 1-12

[13] Zhao M, Zhang M and Jianzhong X 2017 Numerical simulation of flow characteristics behind the aerodynamic performances on an airfoil with leading edge protuberances *Engineering Applications of Computational Fluid Mechanics* 11 pp 193-209

[14] Abbas M K and Hassan H A 2013 Numerical and Experimental study of the effect of winglet at subsonic flow *Al Anbar* 6