ZIBELINE INTERNATIONAL
ISSN: 1024-1752 (Print)
CODEN: JERDFO

Journal of Mechanical Engineering Research & Developments (JMERD)

DOI : http://doi.org/10.26480/jmerd.01.2019.26.33

CrossMark

RESEARCH ARTICLE

DRAG INVESTIGATION AROUND A CYLINDER AND SYMMETRIC AIRFOIL IN TANDEM WITH DIFFERENT GAPS AND ANGLES OF ATTACK

Humam M. Salih, Mustafa A. Abdulhussain

Mechanical Engineering Department, University of Technology, Iraq.
**Corresponding Author Email: humsalih2@gmail.com, mustafa_ahmed2018@yahoo.com*

ARTICLE DETAILS

Article History:

Received 20 November 2018
Accepted 26 December 2018
Available online 22 January 2019

ABSTRACT

A Numerical simulation for the turbulent airflow around a symmetrical airfoil (NACA 0012) adjusted in tandem with circular cylinder, the airfoil were adjusted at variable angle of attack between (5-20°) with positive clockwise inclination setting the cylinder center in front of the leading-edge centroid using the realizable K-ε turbulent model in the ANSYS FLUENT 17 package is performed. The effect of changing the adjusted tandem bodies' configuration, gap (0.05-0.1) and the cylinder size variation on the flow pressure distribution and the airfoil drag coefficient were investigated. The simulation considered in free stream uniform velocity of (25 m/sec) at standard atmospheric pressure. The results showed that using a cylinder with diameter equal to leading edge diameter causes more pressure drop as the tandem gap decreased, noticing that this effect becomes more significant at a high angle of attack. A valuable reduction in drag coefficient of this configuration is achieved and it becomes more with a smaller cylinder diameter with respect to gap distance.

KEYWORDS

symmetric airfoil, tandem cylinder, variable gap, Drag evaluation

1. INTRODUCTION

The tandem gap of Bluff bodies in certain conditions contributes the flow field drag reduction, this reduction depends on several parameters such as the flow velocity range, the bodies' configuration, and their tandem adjustment [1]. These criteria will specify the influence of the exerted forces and moments (including the drag force) on the bodies.

One of the common airfoils drag reduction methods is the insertion of a small circular cylinder in front of it having a size equal to or less than the leading-edge radius that is linearly-positioned with the airfoil chord line. This approach with certain flow conditions and the specified gap distance between the bodies will minimize the exerting forces and moments, including the drag force to a certain value depending on the airfoil angle on attack value and direction since it affects the drag pressure-fraction adversely.

Several types of research have been conducted on the drag reduction for several types of airfoils. Hussain carried out experimental testing of the boundary layer separation position when installing circular cylinder in the front of NACA 0012 leading edge airfoil exposed to high subsonic free stream velocity [2]. The investigation included the variation of the airfoil angle of attack from (0-20°), the cylinder is placed linearly with the chord line in a (1mm) gap in front of the airfoil having a radius greater than the L.E. radius. The results showed that when installing the cylinder, the separation angle of attack has advanced from (14°) to (20°). Also, the lift and drag coefficient variation with increased angle of attack becomes very capricious.

tandem circular airfoils with variable distance spacing by adopting several flow modes; two symmetric vortices, two asymmetric vortices, and single vortex behind the first airfoil using the LS-STAG method for Reynolds number range between (50-300) [3]. For each flow mode, the second airfoil drag coefficient is negative for small gaps and gradually becomes positive, but it remains less than the drag coefficient for the single airfoil.

Some researchers proposed inserting a micro-scale cylinder in front of the leading-edge curvature of wind turbine blade at stall condition as an effective method for regulating the flow separation [4]. They simulated the case using the shear stress transport K-ω turbulence model with the variation of the cylinder size and location, the results indicated that the microcylinder can prevent flow separation without affecting the stability of the turbine and also can increase the blade generated torque by adjusting the appropriate size and position that are relevant to the formed mode of the stall.

The influence of inserting rotating cylinder in the front of the airfoil on the Aerodynamics effectiveness has been carried out numerically by Abdulla and Hamoud, who studied the NACA 0012 symmetric unconventional variable angle of attack airfoil interfaced with rotational cylinder spaced horizontally at variable distances ranging from (0.1-0.8) from the chord length using the SST K- ω turbulence model considering the cylinder to the mainstream velocity ratio within (1-4) [5]. The simulation results indicated that the optimum cylinder/mainstream velocity ratio is (4) with two positioning gaps of (0.5) & (0.8) from the chord length, respectively. In addition, the rotating cylinder has affected the stalled airfoil by reducing the separated region and decreasing the drag coefficient for the airfoil at the same angle of attack.