Computational Study of NACA0012 Using Micro Cylinder at Low Reynolds Number

Mohd. Umar[1], Faisal Shameem[2]

Department of Mechanical Engineering, Galgotias University, Greater Noida, Uttar Pradesh 203201, India

Corresponding author's e-mail: umar.mohd1223@gmail.com

Abstract: **To achieve aerodynamic efficiency, the effect of passive flow plays a vital role and has been the area of interest for research. Due to small modifications in geometry the improvement in performance can be observed. The Computational study on NACA0012 was performed at low reynolds number by placing micro cylinder near its leading edge and investigating the results of lift and drag coefficients at different angle of attack.**

Keywords: **Micro Aerial Vehicle, Airfoil, Viscosity, Micro-cylinder, Reynolds Number**

1. Introduction

Aerodynamic performance of the airfoil has been the interest of research for decades. Sustain research has helped in increasing the efficiency of an airfoil through the development of different types of airfoil. Recent studies in low Reynolds number airfoils have aided military and civilians through various applications like unmanned aerial vehicle(UAV), micro Aerial vehicle (MAV), sailplanes, etc. The condition of the boundary layer and the behaviour of boundary layer separation are critical factors for an airfoil operating at low Reynolds numbers. A modest negative pressure gradient can cause laminar separation bubbles to develop on an airfoil. With reducing speed or incident increase, the short bubble burst into a long bubble/unattached free shear layer. The efficiency of the conventional airfoil at low Reynolds number decreases, leading to efficiencies far less than any of the flat plate airfoils.

When the adverse pressure gradient was adequate, as a turbulent boundary layer, the separated laminar boundary layer can reattach to the upper surface of the airfoil. This causes a laminar separation bubble to develop on the airfoil. The laminar separation bubble that forms on the airfoil gets thinner and shorter as the chord Reynolds number increases. The bubble grows in length and thickness as the angle of attack increases. The laminar separation bubble bursts when the unfavourable pressure gradient gets substantial, causing the airfoil to stall. In order to increase the aerodynamic performance various active and passive ways are used to energize the flow around the airfoil, like implementing slot passage to the airfoil increases efficiency or presence of micro cylinder near the leading edge significantly delays the stall phenomenon. This work presents a computer research that aims to improve the lift and performance of the NACA0012 airfoil by locating a micro cylinder at a low Reynolds number near the leading edge of the airfoil.

2. Literature Review

In an experimental investigation for laminar separation bubble characteristic, M. M. O'Meara et al [1] found that LSB decreases as chord reynold number increases and with the increase in angle of attack both the size and bubble length increases. As Reynolds number increases, the boundary layer starts to split due to the unfavourable pressure gradient on the airfoil surface. An LSB forms at the airfoil when there is a significant enough unfavourable pressure gradient for the separated boundary layer to reconnect itself as a turbulent boundary layer. The LSB was found to burst as adverse pressure gradient becomes too high [2]. Michael V. Ol et al [3] conducted a comparison experiment of LSB in three different apparatus, a low-turbulence wind tunnel, a water tunnel, and a tow tank shows almost qualitative similar results in bubble shape and velocity fields, as well as Reynolds stress distributions. The vortices are created as a result of the detached shear layer's inviscid instability, and the shedding frequency is consistent with linear stability analysis [4], Dilek Funda Kurtulus [5], the shedding time of Karman vortices lengthens as the shedding frequency decreases under stable external conditions at 1000 Reynolds number with larger angles of attack. Serhiy Yarusevych et al [6] conducted study on boundary layer separation and wake development and found that as reynolds number increases the boundary layer reattachment and bubble formation takes place but boundary layer fails to reattach and lead to the formation of wide wake at low reynolds number. Lift force and near wake of an airfoil at ultra-low reynolds number depends upon the AoA and correlate with lift coefficient, the drop in lift coefficient leads to the stall condition [7], behavior and characteristic of the coherent structure in separated shear layer and airfoil wake depends on reynold number and the flow regime, and wake vortex shedding linearly depend on the reynolds number [8].

Shigeru Sunada et al [9] studied at low reynolds number aerodynamics characteristics, by varying its thickness, camber, and roughness of the airfoil and observed that thinner airfoil gives good performance at low reynolds number. As airfoil thickness increases at greater angles of attack, flow separation is delayed and the lift to drag ratio drops at low Reynolds numbers [10].

In order to increase aerodynamic efficiency, several active and passive methods are used, Yue Liu et al [11]) implemented airfoil with grooves over the surface, the grooves help in eliminating LSB and unsteady vortex shedding was found with wider grooves. Lance W. Traub et al [12] conducted test at reynolds number 40000, 60000 and 80000 and the result shows that with the implementation of leading edge and trailing edge can significantly improve airfoil efficiency. maximum lift coefficient increases in both smart flap with smooth and sinusoidal leading edge wing, stall point advances in smart flap with smooth leading edge while delays for sinusoidal leading edge [13]. P W Bearman and D M Trueman [14], conducted an experiment on flow around cylinder and observed that introducing splitter plate into the wake, the increased drag effect can be reduced effectively, and high drag is related to vortex shedding. A computational study of flow over NACA 4415 with a micro cylinder placed near a leading edge was conducted and results show that micro cylinder energizes the flow field on upper surface of the airfoil and significantly delays the stall phenomenon. The effect of oscillating micro cylinder near the leading edge at optimal oscillate method can increase lift-to-drag of airfoil by 88.21%. Researchers tested an airfoil, and the findings showed that the distance between the airfoil's leading edge and the micro cylinder is crucial for preventing stall phenomena and increasing lift drag efficiency; if the distance is (s/c =

0.005), severe blockage happens. The obstruction is greatly relieved above s/c = 0.015 when spacing is increased.

Methodology

The Navier-Stokes equations for incompressible, laminar, two-dimensional flow are the governing equations:

$$\vec{\nabla}.\vec{V} = 0$$

$$\frac{\partial \vec{V}}{\partial t} + (\vec{V}.\vec{\nabla})\vec{V} = -\frac{1}{\rho}\vec{\nabla} p + \nu \nabla^2 \vec{V}$$

Where,

ρ = density
V = velocity
p = pressure
v = viscosity

ANSYS Fluent is used for the simulation of NACA0012 airfoil with the chord length(c) of 0.1m. The airfoil lies at the centre of a semicircle of radius 20c, which serves as the computational domain for the simulation. A subdomain with a width of 40c is rectangular in shape and is attached to the right border of this semicircle. The above dimension for domain is sufficient for aerodynamic coefficient to cause any measurable effects. The left boundary is protected by the velocity inlet boundary condition, the right boundary is protected as a pressure outlet, and the top and bottom borders are protected as a slip wall to prevent the development of a viscous boundary layer along them. The airfoil surface is protected as a non-slip wall, however.

The micro cylinder of 1mm is used and placed near the leading edge of airfoil at 5mm to the negative x-axis and 5mm to the positive y-axis.

The SIMPLE-type totally implicit method is used to accomplish the pressure-velocity coupling. A laminar model is used to simulate the flow in a transient unstable framework.

The fluid medium utilised is air, with default values for its density and dynamic viscosity.

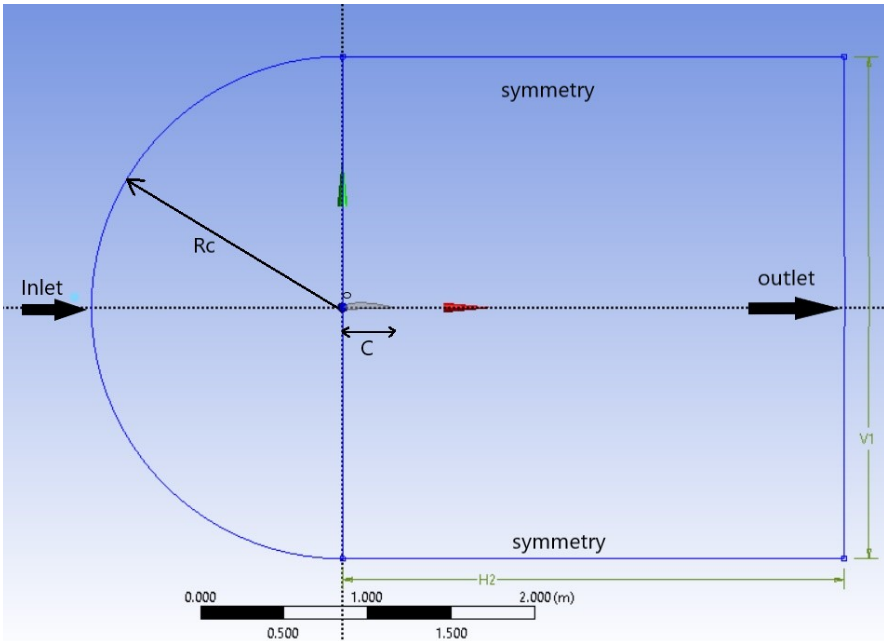

Fig. 1 Schematic diagram of airfoil with cylinder

3.1 Mesh statistics

C-type domain and Hexagonal mesh generation for airfoil is done in this study with the help of ICEM-CFD. There are 200 nodes in the coarse mesh, 300 in the medium mesh, and 500 in the fine mesh. For coarse, medium, and fine mesh, the initial cell spacing between nodes is 0.002c, and for fine mesh, it is 0.0015c. The domain's boundary conditions are defined as solid wall with no-slip conditions for the airfoil boundary, and velocity-inlet and pressure-outlet are assigned, respectively, to the domain's inlet and outlet.

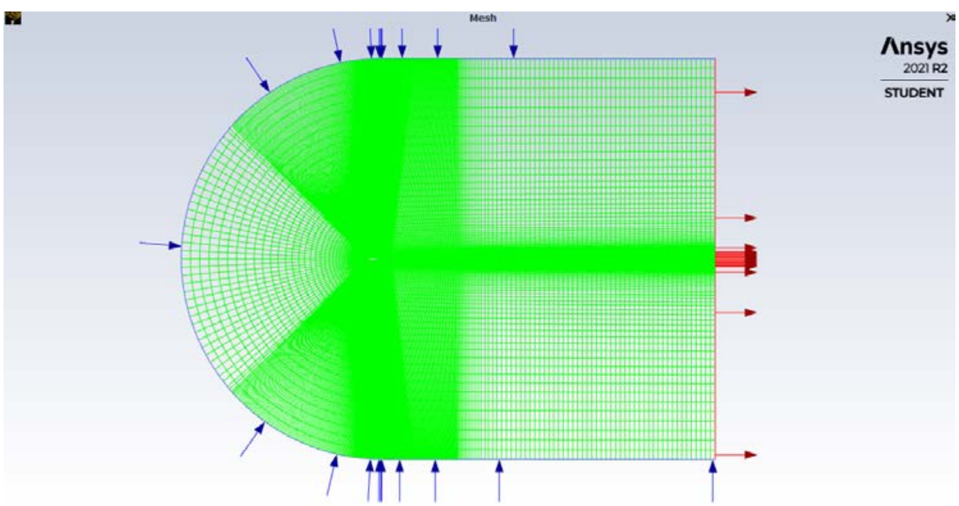

Fig. 2 mesh Domain for NACA0012

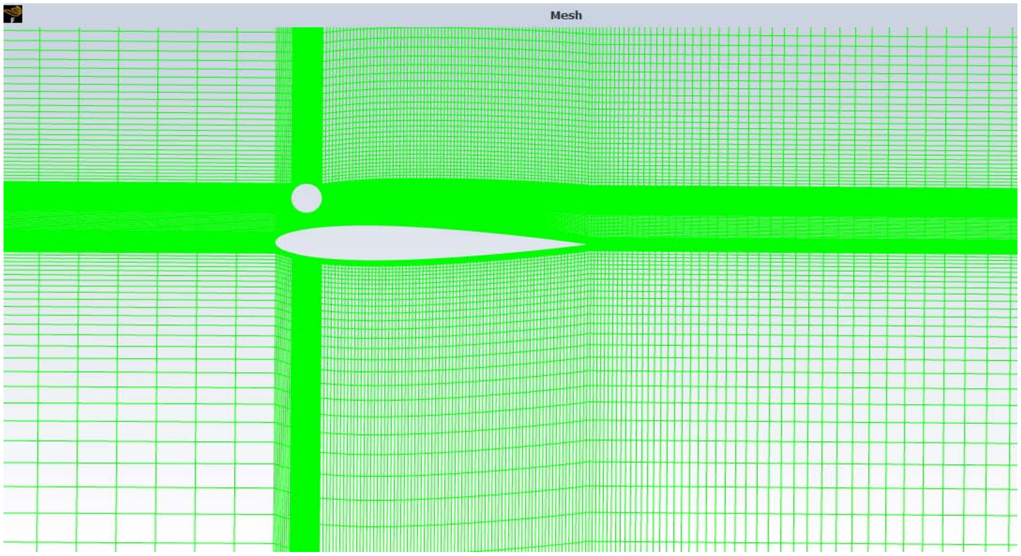

Fig. 3 Mesh domain near NACA0012 airfoil with cylinder

3.2 Grid Independence test

In order to predict the accurate result for the study, mesh quality inspection or mesh validation is done using a grid independence test.

Mesh type	No. of elements	Lift coefficient	Drag coefficient
coarse	40859	0.358	0.0903
medium	81563	0.358	0.0904
fine	159656	0.359	0.0900

Table1 Value of lift and drag coefficient w.r.t. Mesh type

Richardson extrapolation theory states that the refinement can't be less than 1.3.
Refinement ratio 1= fine mesh/medium mesh = 1.957
Refinement ratio 2= medium mesh/coarse mesh = 1.996
These result confirms that the generated mesh are in optimum condition.

3.3 Data Validation

Data validation is done in order to justify boundary domain and conditions, on the basis of lift coefficient at different angles of attacks and comparing with data at Re=1000 calculated by Kurtulus et al. and Liu et al. The data in present study is close to published data with an error less than 5%.

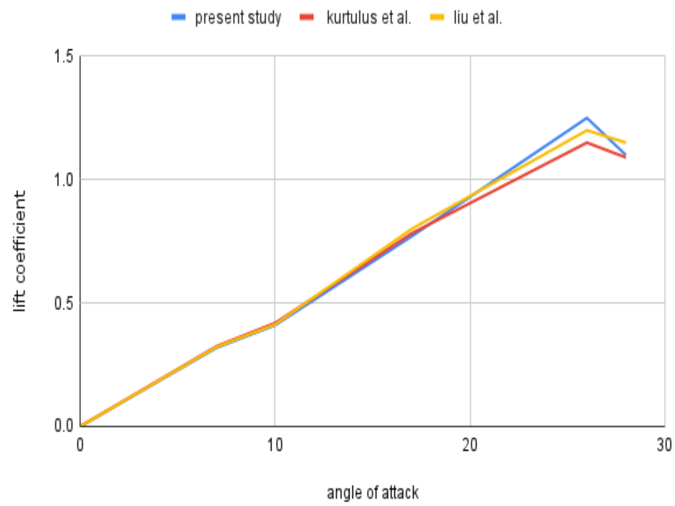

Fig. 4 Lift coefficient w.r.t. the angle of attack, at Re = 1000

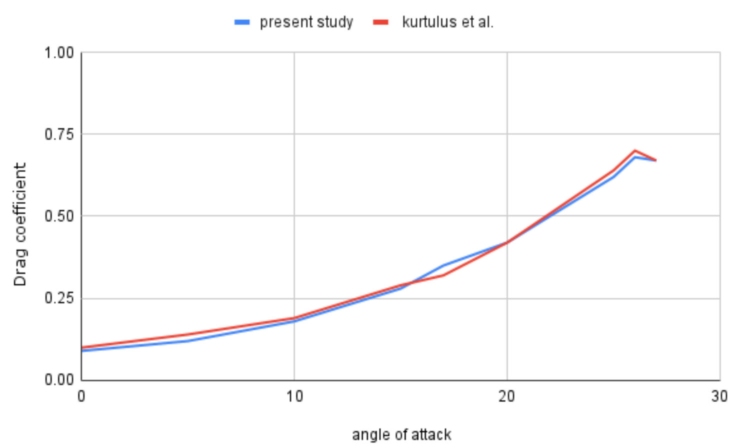

Fig. 5 Dragcoefficient w.r.t the angle of attack, at Re = 1000

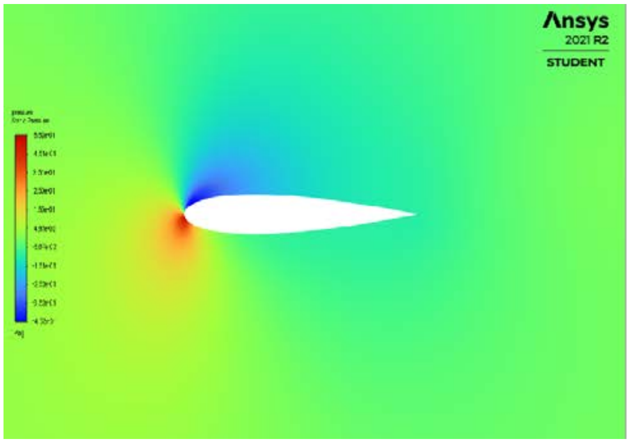

Fig. 6a pressure velocity contour at AoA=7 degree

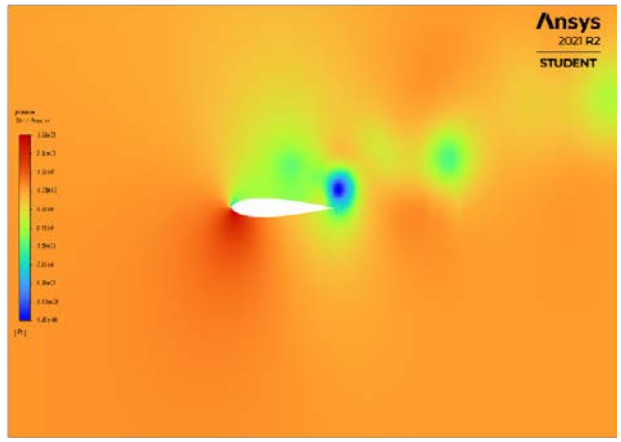

Fig. 6b pressure velocity contour at AoA=19 degree

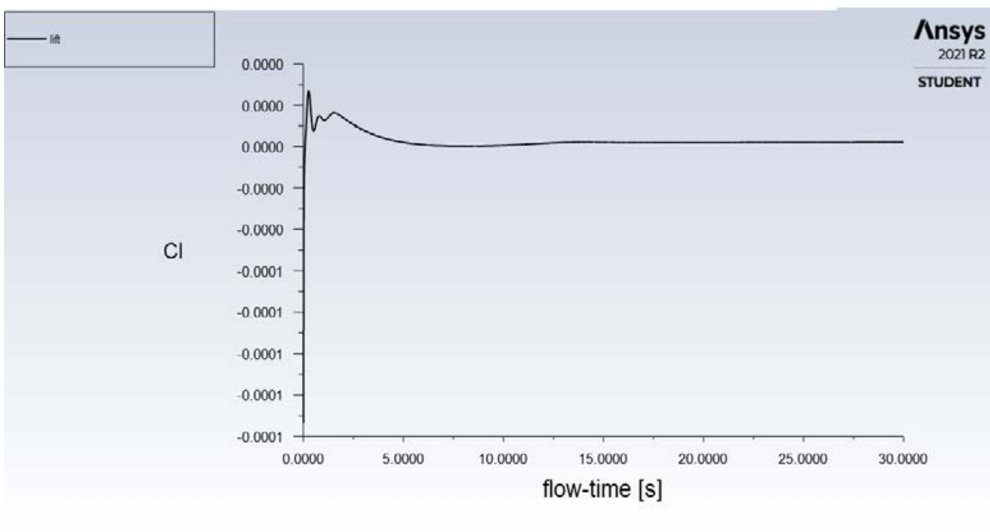

Fig. 7a Lift coefficient as a function of flow-time at AoA= 0degree

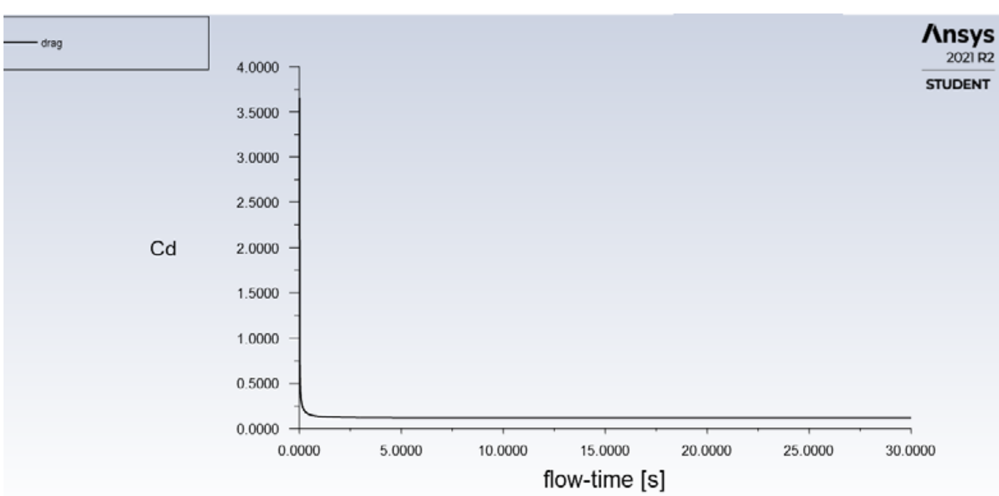

Fig.7b Drag coefficient as a function of flow-time at AoA= 0degree

3. Conclusion

Research has focused on the effect of passive flow, which is crucial to achieving aerodynamic efficiency. Performance has improved as a result of minor changes to the geometry. Using a micro cylinder placed near the edge of NACA0012, a computational analysis was conducted at lower Reynolds number to examine effects of lift and drag coefficients at various angles of attack.

A grid independence test is used for mesh quality inspection or mesh validation in order to accurately forecast the study's outcome and it is found that the generated mesh is in optimum condition. Validating the data is another step in justifying the boundary domain and conditions. This is done by comparing the data at 1000 reynolds number, which was estimated by Kurtulus et al. and Liu et al., with the lift coefficient at various angles of attack. The current study's data, with an error of less than 5%, is fairly close to published data.

References

1. M. M. O'Meara and T. J. Mueller "Laminar Separation Bubble Characteristics on an Airfoil at Low Reynolds Numbers" AIAA JOURNAL VOL. 25, NO. 8, AUGUST 1987
2. Hui Hu and Zifeng Yang "An Experimental Study of the Laminar Flow Separation on a Low-Reynolds-Number Airfoil" Journal of Fluids Engineering, MAY 2008, Vol. 130.
3. Michael V. Ol, Brian R. McAuliffe, Ernest S. Hanff, Ulrich Scholz and Christian Kahler "Comparison of Laminar Separation Bubble Measurements on a Low Reynolds Number Airfoil in Three Facilities" 35th AIAA Fluid Dynamics Conference and Exhibit 6 - 9 June 2005, Toronto, Ontario Canada.
4. Laura L. Pauley, Parviz Moinz and William c. Reynolds "The structure of two-dimensional separation" J. Fluid Mech. (1990), vol. 220, pp. 397411
5. Dilek Funda Kurtulus "On the Unsteady Behavior of the Flow Around NACA 0012 Airfoil with Steady External Conditions at Re=1000" International Journal of Micro Air Vehicles, 2015 - journals.sagepub.com.
6. Serhiy Yarusevych and Pierre E. Sullivan "Investigation of Airfoil Boundary Layer and Wake Development at Low Reynolds Numbers" AIAA Fluid Dynamics Conference and Exhibit 28 June - 1 July 2004, Portland, Oregon
7. Md. Mahbub Alam, Y. Zhou, H. X. Yang, H. Guo and J. Mi "The ultra-low Reynolds number airfoil wake" Exp Fluids (2010) 48:81–103 DOI 10.1007/s00348-009-0713-7.
8. Serhiy Yarusevych, Pierre E. Sullivan and John G. Kawall "On vortex shedding from an airfoil in low-Reynolds-number flows" J. Fluid Mech. (2009), vol. 632, pp. 245–271.
9. Shigeru Sunada, Akitoshi Sakaguchi and Keiji Kawachi "Airfoil Section Characteristics at a Low Reynolds Number" Journal of Fluids Engineering, March 1997, Vol. 119/129.
10. Emad Uddin, Muhammad Adil Naseem, Saif Ullah Khalid, Aamir Mubashar and Samiur Rehman Shah "Investigation of the flow around uncambered airfoils at 1000 Reynolds number using computational fluid dynamics for micro air vehicles" The 2017 World Congress on Advances in Structural Engineering and Mechanics at: Seoul, South Korea
11. Yue Liu, Peifeng Li, Wei He and Kaiyong Jiang "Numerical study of the effect of surface grooves on the aerodynamic performance of a NACA 4415 airfoil for small wind turbines" Journal of Wind Engineering & Industrial Aerodynamics 206 (2020) 104263, https://doi.org/10.1016/j.jweia.2020.104263.
12. Lance W. Traub and Cory Coffman "Efficient Low-Reynolds-Number Airfoils" JOURNAL OF AIRCRAFT Vol. 56, No. 5, September–October 2019
13. AA Mehraban and MH Djavareshkian "Experimental investigation of ground effects on aerodynamics of sinusoidal leading-edge wings"Journal of Mechanical Engineering Science 0(0) 1–14 http://dx.doi.org/10.1177/0954406220975422

14. P W Bearman and D M Trueman "An Investigation of the Flow around Rectangular Cylinders" The Aeronautical Quarterly, 1972 - cambridge.org https://doi.org/10.1017/S0001925900006119