Journal of Basic and Applied Engineering Research
p-ISSN: 2350-0077; e-ISSN: 2350-0255; Volume 4, Issue 6; July-September, 2017, pp. 517-519

http://www.krishisanskriti.org/Publication.html

Numerical Simulation of Flow over an Aerofoil on Wind Tunnel Environment

Sombuddha Bagchi[1], Pringale Kumar Das[2], Soham Mondal[3]
Sourav Sarkar[4] and Pranibesh Mandal[5]

[1,2,3]Mech. Engg. Department, Jadavpur University Kolkata-700032
[4,5]Mechanical Engineering Department, Jadavpur University Kolkata-700032
E-mail: [1]sombuddha.bagchi@gmail.com, [2]rikd37@gmail.com, [3]shmistrulyhot63@gmail.com,
[4]souravsarkar.iitm@gmail.com, [5]m.pran11@gmail.com

Abstract—*Aerofoils find extensive use in development of numerous air operated machineries. Experimental aerodynamic studies in wind tunnels to determine the pressure distribution around an aerofoilhavebecomea regular practice, further contributing in determining the lift coefficient generated on it. However, experimental studies involve huge cost reducing the possibility of observing over a wide range of situation seven if it has given ideas about the exact phenomenon. CFD based simulations have been adopted with significant accuracy levels for the aforesaid problem which not only allow for a wider and more intricate ranges of observation but also lays the foundation of the hypotheses for a good experimental study. The present paper deals with the numerical simulations of flow over a NACA3119 aerofoil at negative angles of attack in environment of a wind tunnel to achieve the pressure and velocity distributions and to determine the lift coefficient generated on it. The results obtained from this numerical simulation agree closely to the theory of fluid mechanics.*

Keywords*: Simulation, Wind Tunnel, Aerofoil.*

1. INTRODUCTION

Aerofoils have been an important topic of study in the field of aerodynamics. It is used in development of aircrafts[1],wind mills[2], wind turbines[3]and such machines which operates due to flow of air. Another important tool used in this field is the Wind Tunnel. It's used for various experimental aerodynamic studies such as pressure contours, velocity profile, streamline around an object as well as drag and lift co-efficient generated on an object. Among all these pressure distributions around an aerofoil is the main source of lift generation and depends on many factors like the shape and size of the aerofoil [4], angle of attack [5], free stream velocity [6] and Mach Number [7]. However, experimental studies despite being much accurate and accepted involve costly instruments and experimental set ups. So, now-a-day computational techniques have grown in stature to replace experimental studies with considerable accuracy. Physically performing an experiment or making a product comes with its own limitations. Due to the same, it is not always possible to check every intricate detail or reach every minute location. Under such conditions simulation comes as a handy tool. Owing to the present computing power in public domain, simulation has come up as one of the predominant steps in development of any new product or in performing any experiment. Simulation is an easy and quick way of determining any faults or mistake in the product or experiment that might have been overlooked in the initial stages. It also helps its user to visualize the product or experiment before it takes physical form and hence helps in savingtime, cost and energy.

The current study incorporates the numerical simulation of airflow over an asymmetric aerofoil placed in a closed-circuit wind tunnel. The results produced are from a simulation which has been performed on ANSYS FLUENT 14.5 software. The main objectives of the present study have been comparison of pressure contours, velocity profile, streamlines and determination of drag and lift coefficient generated on an aerofoilfornegative angles of attack on a NACA3119 aerofoil which is not among the standard aerofoils. The analysis of drag and lift co-efficient for negative angles of attack are seldom performed but is an important factor as far as tail life generation of an airplane is concerned.

2. PROCEDURE:

2.1 Problem Geometry and Governing Equations:

A rectangular flow domain (4m x 1m) is used to simulate a wind tunnel and an aerofoil section is placed within it. The A wooden asymmetric aerofoil, schematically shown in Fig. 1, has been used with 21.9cm chord length, the leading-edge radius and maximum thickness being 2.1cm and 4.2cm respectively. The maximum camber is 0.7cm occurring at 10% of chord length, at the point A shown below. As per NACA 4-digit nomenclature, this particular aerofoil section can be called a NACA3119, which is obviously not among the standard aerofoil. For the present study, the aerofoil has been set at -10° and -20° angle of attack.

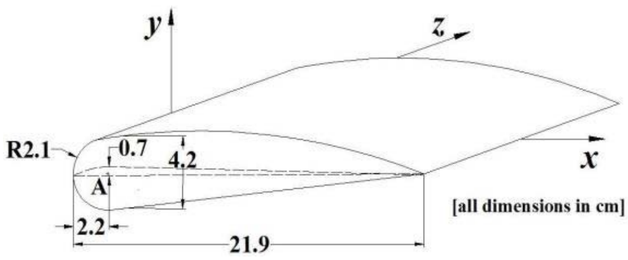

Fig. 1: Aerofoil Geometry

The well-known Continuity, Momentum and Bernoulli's Equations have been used for performing the simulation which are given respectively by,

$$\frac{\partial \rho}{\partial t} + \nabla.(\rho\vec{V}) = 0 \qquad (1)$$

$$\frac{\partial(\rho\vec{V})}{\partial t} + \nabla.(\rho\vec{V}\vec{V}) = -\nabla p + \nabla.\vec{\tau} + \rho\vec{g}(2)$$

where p is the static pressure, $\vec{\tau}$is the stress tensor,$\rho\vec{g}$ is the gravitational body force and$\vec{F}$ is any other external body force such as that arising from interaction with the discrete phase.

The stress tensor is given by

$$\vec{\tau}=\mu\left[(\nabla\vec{V} + \nabla\vec{V}^{T}) - \frac{2}{3}(\vec{V}\,I)\right](3)$$

while the Lift and Drag coefficient is given by,

$$C_L = \frac{F_L}{\frac{1}{2}\rho U_\infty^2 D}(4)$$

$$C_D = \frac{F_D}{\frac{1}{2}\rho U_\infty^2 D} \qquad (5)$$

2.2 Numerical Method and Boundary Conditions:

A finite volume based CFD code ANSYS Fluent 14.5 has been used to perform the required numerical simulations. A pressure-based solver is chosen as the numerical scheme and a steady analysis has been performed. A viscous and turbulent standard *k-ε* 2-equation model has been used with standard wall functions and curvature corrections. The model constants are C_{mu}= 0.09; C_2-ε= 1.92; C_1- ε= 1.44; *TKEPrandtl no.*= 1 and *TDRPrandtl no.*= 1.3. The solution method is based on the SIMPLE scheme which provides better and faster convergence results at steady turbulent models. A least squares cell based gradient has been used and the momentum, Turbulent Kinetic energy and Turbulence Dissipation rate has been solved using the second order upwind scheme for more accurate results. The inlet conditions of the model were set to velocity-inlet with a free stream velocity of 20m/s. The turbulent intensity was limited to 5% and the turbulent viscosity ratio set to 10. The upper and lower walls of the wind tunnel geometry was set to no-slip wall condition. The outlet conditions of the model were set to outflow to simulate the closed-circuit nature of the wind tunnel. The Lift and Drag coefficients were monitored on the aerofoil. The Standard initialization has been done from inlet and the solution was converged within 1048 iterations with the total calculation duration of 20 minutes.

2.3 Mesh Details

The ANSYS Meshing package was used to create an all Triangles based unstructured grid as shown in the Fig. 2.The maximum size and face size of element was limited to 0.05m. The defeaturing tolerance was reduced to 1.e-005m. For more accurate and better calculations, an inflation was performed around the aerofoil geometry. It had a final layer height of 1.e-05m and maximum number of layers 50 with a growth rate of 1.5. The total number of nodes and elements were 2,40,915 and 2,45,296 respectively.

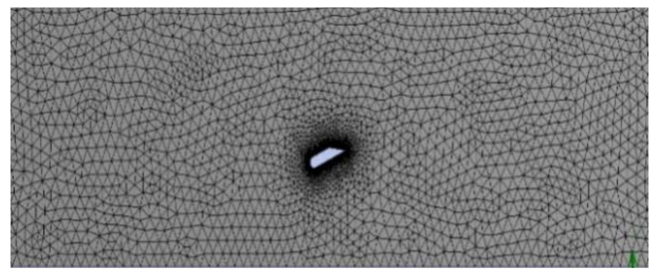

Fig. 2: Mesh Details

3. RESULTS AND DISCUSSIONS:

The contours delineate the pressure and velocity with the maximum value being represented by red while the minimum value is in blue.

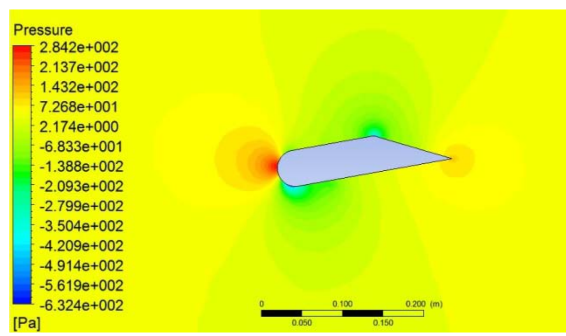

Fig. 3.1. Pressure contours at -10° angle of attack

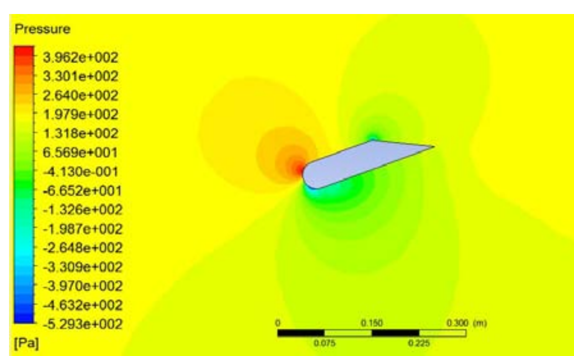

Fig. 3.2. Pressurecontours at -20° angle of attack

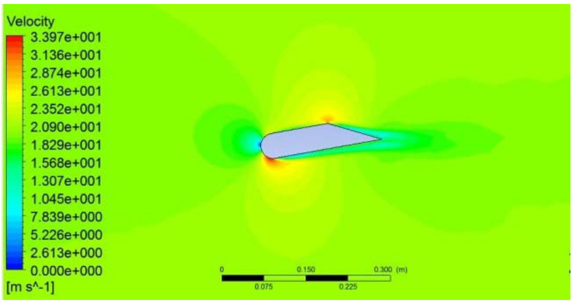

Fig. 3.3. Velocity contours at -10° angle of attack

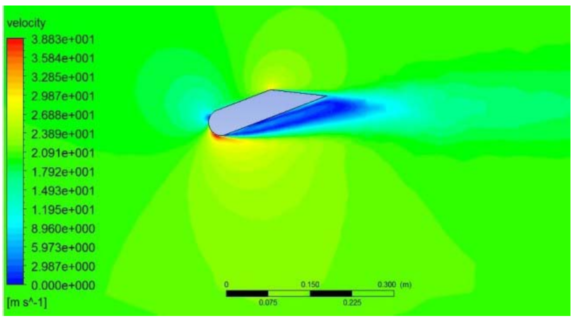

Fig. 3.4. Velocity contours at -20° angle of attack

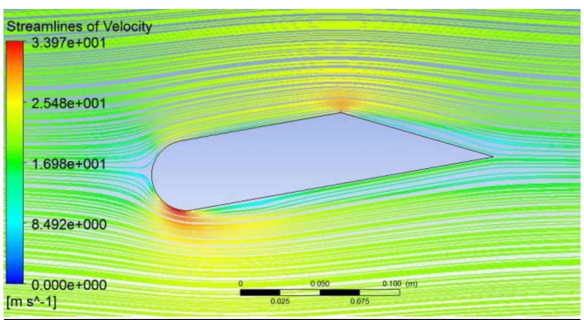

Fig. 3.5. Velocity streamlines at -10° angle of attack

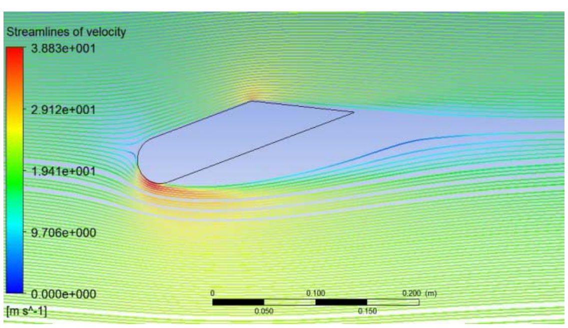

Fig. 3.6. Velocity streamlines at -20° angle of attack

For -10° angle of attack, a stagnation point is observed around the leading edge of the aerofoil. The Bernoulli's principle holds true; we observe a high pressure around that region. Due to the same principle, we observe a high velocity just below the leading edge and on the upper surface of the aerofoil and a corresponding low-pressure region.

For -20° angle of attack the situation changes drastically. A comparatively large number of stagnation points are observed on the lower surface of the aerofoil and hence, a corresponding high-pressure region. A small region of low pressure or high velocity is observed on the lower point of leading edge.

Table 1: Lift and Drag coefficients

Angle of attack	Lift coefficient	Drag coefficient
-10°	-0.733	5.282
-20°	-50.97	17.37

The magnitude of negative lift coefficient is observed to increase byalmost a factor of 70 for anincrease in negative angle of attack from -10° to -20°. Simultaneously there is also an increase in the drag coefficient by a factor of 3 for the above-mentioned increase in the negative angles of attack.

4. CONCLUSION:

Numerical analyses of flow over an aerofoil in a wind tunnel environment for negative angles of attack have been performed. Negative lift coefficient has been observed, magnitude of which increases with increase in the negative angle of attack. The drag coefficient has also been observed to increase with increase in the negative angle of attack.

5. FUTURE SCOPE OF WORK:

The numerical simulation that has been performed will act as a standard for further experimental studies in Closed-Circuit Wind Tunnels for aerofoils placed at negative angles of attack.

REFERENCES:

[1] Selig, M.S., Maughmer, M.D. and Somers, D.M., 1995. Natural-laminar-flow airfoil for general-aviation applications. *Journal of aircraft*, *32*(4), pp.710-715.

[2] Mojola, O.O., 1985. On the aerodynamic design of the Savonius windmill rotor. *Journal of Wind Engineering and Industrial Aerodynamics*, *21*(2), pp.223-231.

[3] Yan, L.W., Ai, C.J. and Xie, H., 2013. New Wind Turbine High-Speed Shaft Design and Simulation of Hydraulic Shock Absorbers. In *Applied Mechanics and Materials* (Vol. 397, pp. 479-482). Trans Tech Publications.

[4] Roberts, W.B., 1980. Calculation of Laminar Separation Bubbles and Their Effect on Airfoil Performance. *AIAA journal*, *18*(1), pp.25-31.

[5] Timmer, W.A. and Van Rooij, R.P.J.O.M., 2003. Summary of the Delft University wind turbine dedicated airfoils. *Transactions-American Society of Mechanical Engineers journal of solar energy engineering*, *125*(4), pp.488-496.

[6] Sherif, S.A. and Pletcher, R.H., 1989. Measurements of the flow and turbulence characteristics of round jets in crossflow. *Journal of Fluids Engineering*, *111*(2), pp.165-171.

[7] SCHNERR, G. and Dohrmann, U., 1990. Transonic flow around airfoils with relaxation and energy supply by homogeneous condensation. *AIAA journal*, *28*(7), pp.1187-1193