DETERMINATION OF PRESSURE COEFFICIENT AROUND NACA AIRFOIL

Ashish P Kunjumon  
UG Student, Department of Mechanical Engineering,  
Mar Baselios Christian College of Engineering and technology  
Kuttikanam, Peermade

Felix Mathew Koshy  
UG Student, Department of Mechanical Engineering,  
Mar Baselios Christian College of Engineering and technology  
Kuttikanam, Peermade

Steevo Tom Jacob  
UG Student, Department of Mechanical Engineering,  
Mar Baselios Christian College of Engineering and technology  
Kuttikanam, Peermade

Sunildutt  
Assistant Professor, Department of Mechanical Engineering,  
Mar Baselios Christian College of Engineering and technology  
Kuttikanam, Peermade.

Abstract  
The aim of the present study is to emphasize mainly the parameters such as the distribution of pressure and velocity over an airfoil surface. The complete process of modelling and simulation of the airfoil at various angle of attacks is investigated. The project focus mainly on the simulation of the airflow around the airfoil. The fluid flow simulations are obtained with ANSYS Fluent 14.5. The pre-processing including the creation and modification of the surface mesh in ANSYS with two dimensional grid generation. The working fluid used is air. Incompressible inviscid flow, combinations of the angle of attack and inlet velocity are the various parameters.  

Keywords: Airfoil, Angle of Attack, Lift force, Pressure coefficient, Simulation

Introduction  
Aerodynamics is a branch of science that deals with the analysis of flow over a body. An airfoil is the shape of a wing or blade or sail as seen in cross-section. An airfoil-shaped body moved through a fluid produces an aerodynamic force. The component of this force perpendicular to the direction of motion is called lift. The component parallel to the direction of motion is called drag. An airfoil is a streamline body which has a rounded leading edge, is elongated and is given a gradual curvature in the flow direction. By using ANSYS, flow analysis becomes more effective as it investigates everything more thoroughly than experimental method.  
Computational fluid dynamics provides a qualitative and sometimes even quantitative prediction of fluid flow by means of mathematical modelling, numerical method and software tools. CFD analysis enables an engineer to compute the flow numerically in a ‘virtual flow laboratory’. The analysis consists of several steps such as: problem statement, mathematical modelling, mesh generation, space discretization, time discretization, iterative solver, simulation run, post processing, and verification.  
ANSYS is vast computational software that enables researchers to analyse the problems related to different engineering sectors. It is used to solve problems related to heat transfer, fluid flow, turbulence, industrial machineries, explicit dynamics, and structural analysis with the assistance of numerical analysis.  
Narayan U Rathod [1] , carried out to emphasize mainly the parameters such as the distribution of pressure is more in the lower part of the leading edge and distribution of velocity is more in the upper part of the leading edge. Mayank Pawar et al.[2] , study deals with the study of static pressure distribution...
over the surface of an aerofoil. Shivasharanayya Hiremath et al. [3] work presents the simulated flow over an aircraft and observed that the lift increases as angle of attack increases.

MD. Safayet Hossain et al. [4] work presents, flow analysis of two airfoils (NACA 6409 and NACA 4412) was investigated. Drag force, lift force as well as the overall pressure distribution over the airfoils were also analyzed. N. Gregory et al. [5] Results are presented for the aerodynamic characteristics of NACA 0012 airfoil section at two different Reynolds numbers.

Airfoils and aerodynamic shaped objects are extensively used in all types of air vehicles for example space shuttle, aircrafts, helicopters, acrobatic aircrafts, and even in various types of missiles. Besides, when it comes to fluid machineries such as pump, turbine, windmill, the shape of impeller, propeller is very important.

In this investigation, study about the flow characteristics around symmetrical and cambered airfoil at zero angle of incidence was analyzed. The variation of pressure difference on the upper and lower surface of an airfoil at the leading and trailing edge was determined and the velocity flow patterns around the symmetrical and cambered airfoil at different angle of attack was also investigated.

Methodology
Computational Fluid Dynamics (CFD) is a numerical method used to simulate physical problems with use of governing equations. This method can be used to investigate design approaches without creating a physical model and can be a valuable tool to understand conceptual properties of new mechanical designs. By using a simulation instead of doing lab experiments, one may acquire results faster and with less expense. In this paper, NACA 0012 and NACA 2424, the well documented airfoils from the 4-digit series of NACA airfoils are utilized. The NACA 0012 airfoil is symmetrical, and NACA 2424 is cambered. Maximum thickness 12% at 30% chord. Maximum camber 0% at 0% chord for NACA 0012. Max thickness 24% at 30% chord. Max camber 2% at 40% chord for NACA 2424. Reynolds number for the simulations was Re = 3 × 10^5.

Following is the method employed to carry through the CFD simulation:

1. Preparing geometric model.
2. Generate meshing.
3. Setting boundary conditions.
4. Software (Fluent) setup, initialization.

Preparing geometric model

The airfoil geometries was acquired as co-ordinate vertices i.e. texts file are imported into the ANSYS FLUENT. FLUENT is essential in the process of doing the CFD analysis, it creates the working environment where the object is simulated. An important part in this is creating the mesh surrounding the object. The mesh and edges must also be grouped in order to set the necessary boundary conditions effectively.

Firstly, airfoil co-ordinates are imported and the curve is formed. The 2D analysis type is used and launch the design model created. Then create the surface to the curve, then the airfoil is generated. Next step is to create a surface from this sketch. The final step of creating the C-Mesh is creating a surface between the boundary and the airfoil by using Boolean operations. In the final step of creating the geometry, the new surface is splitted into 4 quadrants and divided into 5 blocks. The geometry is finished. Save the project and close the design modeler. Create the mesh for the simulation.

Fig 1. Physical domain of the problem

Fig 2. Dividing the C-Mesh into 5 blocks and 4 quadrants.
Generate Mesh
An environment consisting of 2 squares and 1 semicircle surrounds the airfoil. For the airfoil a structured quadratic mesh is used with a fine mesh around the airfoil. A fine mesh implies a higher number of calculations which in turn makes the simulation use longer time to finish.

Grid convergence: Two different meshes are made to the CFD simulations. This is to test the grid convergence, how the change in number of cells, and hence also cell size, affect the result. Grid independence study is conducted by using successively smaller cell sizes.

<table>
<thead>
<tr>
<th>Table 1: Grid Independence Study NACA0012</th>
</tr>
</thead>
<tbody>
<tr>
<td>MESH</td>
</tr>
<tr>
<td>---------</td>
</tr>
<tr>
<td>MESH A</td>
</tr>
<tr>
<td>MESH B</td>
</tr>
<tr>
<td>MESH C</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Table 2: Grid independence study NACA 2424</th>
</tr>
</thead>
<tbody>
<tr>
<td>MESH</td>
</tr>
<tr>
<td>---------</td>
</tr>
<tr>
<td>MESH A</td>
</tr>
<tr>
<td>MESH B</td>
</tr>
<tr>
<td>MESH C</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Table 3: Operating parameters</th>
</tr>
</thead>
<tbody>
<tr>
<td>INPUT</td>
</tr>
<tr>
<td>Velocity</td>
</tr>
<tr>
<td>Operating temp</td>
</tr>
<tr>
<td>Operating pressure</td>
</tr>
<tr>
<td>Models</td>
</tr>
<tr>
<td>Density of fluid</td>
</tr>
<tr>
<td>Kinematic viscosity</td>
</tr>
<tr>
<td>Angle of attack</td>
</tr>
<tr>
<td>Fluid</td>
</tr>
</tbody>
</table>

Fig 4. Mesh around NACA 0012 airfoil

Fig 5. Mesh around NACA 2424 airfoil

Setting Boundary Conditions
Giving properties to the different geometries is vital to make the simulation work. In this case, the mesh boundaries were given set to the x and y velocity components, and the end boundary the property pressure-outlet to simulate the zero gauge pressure, and the rest of the C Mesh is taken as velocity inlet. The airfoil itself is given as wall properties.
Setting up FLUENT (Initializing and solving)

The geometry and mesh were imported into FLUENT. Laminar flow (streamline flow) occurs when a fluid flows in parallel layers. Laminar flow occurs at low Reynolds numbers, where viscous forces are dominant, and is characterized by smooth, constant fluid motion. Here, the flow is laminar and the boundary layer is a laminar layer. Laminar flow is a flow regime characterized by high momentum diffusion and low momentum convection. The density based solver is used. Inviscid laminar flow is used and the fluid is air. In this work instead of tilting the airfoil the free stream air is changing its angle by using cosine component for \( x \)-velocity and sine component for \( y \)-velocity. The boundary condition is applied and computation is done from inlet. Convergence criteria is \( 10^{-6} \). Before calculation it is initialized and we run the calculation.

Governing Equations

The conservative form of governing equations for the steady flow field are refer to (1), (2), (3), (4), (5).

Continuation equation:
\[
\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} = 0
\]  
(1)

Navier-Stokes Equation:
\[
u \frac{\partial u}{\partial x} + \nu \frac{\partial v}{\partial y} = -\frac{\partial P}{\partial x} + \mu \left[ \frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2} \right]
\]  
(2)

Coefficient of Lift:
\[
2L/\rho AV^2
\]  
(3)

Pressure coefficient Cp:
\[
(Cp) = \frac{p - p_e}{\rho AV^2}
\]  
(4)

Bernoulli’s Equation :
\[
P + \frac{1}{2} \rho V^2 = \text{constant}
\]  
(5)

Results and Discussion

For symmetrical airfoil at zero angle of attack the pressure distribution over the upper surface is similar to the pressure distribution on the lower surface, this implies that no lift is generated at zero degree angle of attack. For cambered airfoil at zero degree angle of attack the static pressure is little higher on the lower surface than on the upper surface which creates a lift force. The pressure and velocity distributions are plotted which illustrate the effect of distribution of pressure and velocity over the airfoil surface for different angle of attack in NACA 0012, NACA 2424 series airfoils. The flow accelerates on the upper side of the airfoil and the velocity of flow decreases along the lower side and according to Bernoulli’s theorem the upper surface will experience lower pressure than the lower surface. The distribution of pressure coefficient under different angles of attack is shown in the following figures 17 and 18. The pressure coefficient of the airfoil’s upper surface was negative and the lower surface was positive, thus the lift force of the airfoil is in the upward direction. It is found that larger the attack angle, greater is the difference of pressure coefficient between the lower and upper surface. The coefficient of pressure difference is much larger on the front edge than the rear edge, thus indicates that the lift force of the airfoil is mainly generated from the front edge.

Angle of attack: 0°

![Fig 5. Pressure contour of zero degree Angle of Attack at 5m/s velocity NACA 0012](image)

![Fig 6. Velocity vector of zero degree Angle of Attack at 5m/s velocity NACA 0012](image)
Fig 7. Pressure contour of zero degree Angle of Attack at 5m/s velocity NACA 2424

Fig 8. Velocity vector of zero degree Angle of Attack at 5m/s velocity NACA 2424

Fig 9. Pressure contour of 6 degree Angle of Attack at 5m/s velocity NACA 2424

Fig 10. Velocity vector of 6 degree Angle of Attack at 5m/s velocity NACA 0012

Angle of attack: 6°

Fig 11. Pressure contour of 6 degree Angle of Attack at 5m/s velocity NACA 2424

Fig 12. Velocity vector of 6 degree Angle of Attack at 5m/s velocity NACA 2424

Fig 11. Pressure contour of 6 degree Angle of Attack at 5m/s velocity NACA 2424

Fig 12. Velocity vector of 6 degree Angle of Attack at 5m/s velocity NACA 2424

Fig 9. Pressure contour of 6 degree Angle of Attack at 5m/s velocity NACA 0012
Angle of attack: 9º

Fig 13. Pressure contour of 9 degree Angle of Attack at 5m/s velocity NACA 0012

Fig 14. Velocity vector of 9 degree Angle of Attack at 5m/s velocity NACA 0012

Fig 15. Pressure contour of 9 degree Angle of Attack at 5m/s velocity NACA 242

Fig 16. Velocity vector of 9 degree Angle of Attack at 5m/s velocity NACA 242

Fig 17. Pressure coefficient graph of 6 degree Angle of Attack at 5m/s velocity NACA 0012 & NACA 2424

Fig 18. Pressure coefficient graph of 9 degree Angle of Attack at 5m/s velocity NACA0012 NACA 2424

Table 4: Comparison of Pressure Coefficient

<table>
<thead>
<tr>
<th></th>
<th>Numerical result</th>
<th>Experimental data</th>
</tr>
</thead>
<tbody>
<tr>
<td>Cp</td>
<td>-2.3</td>
<td>-2.5</td>
</tr>
</tbody>
</table>

The experimental configuration of Gregory & O'Reilly et al. [5] has been considered for experimental validation of the symmetrical airfoil numerical study. Above is a table displaying the comparison of Coefficient of Pressure along the airfoil on the upper surface at the front edge for the experimental data and the CFD simulation. From the table it is clear that CFD matches the experimental data fairly well.
Conclusion

This work presents the simulated flow over an airfoil and it was observed that for symmetrical airfoil at zero degree angle of attack the pressure distribution over the upper surface is similar to the pressure distribution on the lower surface and no lift force is generated. For cambered airfoil at zero degree angle of attack the static pressure is little higher on the lower surface than on the upper surface which creates a lift force. For both symmetrical and cambered airfoil it is seen that as the angle of attack increases the coefficient of pressure difference also increases and it is much larger on the leading edge than the trailing edge. From the pressure coefficient graphs, it is seen that as the angle of attack increases the coefficient of pressure difference also increases and it is much larger on the leading edge than the trailing edge. Even though the Angle of Attack is similar for both the airfoils more lift is generated in cambered airfoil so cambered airfoil is more efficient.

Reference


[3] [3]. MD. Safayet Hossain, Muhammad Ferdous Raiyan, Mohammed Nasir Uddin Akanda, Nahed Hassan Jony, A Comparative Flow Analysis Of NACA 6409 AND NACA 4412 Aerofoil, Department of Mechanical Engineering, CUET, Chittagong, Bangladesh, 342-350
