Abstract - Analysis of Effect of flow over NACA 2412 airfoil was done using the available Computational Fluid Dynamics methods. Popular CAE software Ansys CFX was used for analysis and simulation. Firstly, the coordinates of the said airfoil model were obtained from the airfoil database of the University of Illinois, and then the coordinates were imported to the geometry modeler of Ansys. CFX (Fluid Flow) was used to generate mesh and conduct the experimentation. At the end of modeling and meshing, simulation was done to observe the effect of flow at a high Reynolds number, on the said airfoil model with respect to pressure distribution, velocity distribution, various aerodynamic forces while maintaining variation in its angle of attack. To conduct the entire experiment, we did not go for experimental set up as the physical model has to be placed inside a wind tunnel, and this process is quite grueling, time consuming and expensive too at the same time. Moreover, the wind tunnel experiments are subjected to the accuracy of the developed model. With the advancement of high configured computers and computational methods, the flow behaviors of the fluid and its effect can be accurately analyzed.

Keywords - CFD, SST Turbulence Model, Angle of Attack, Stall.

I. INTRODUCTION

The NACA airfoils are the shapes designed for aircraft wings developed by the National Advisory Committee for Aeronautics. The NACA 2412 is a four-digit airfoil series profile. An airfoil generates lift by exerting a downward force on the air as it flows past. When oriented at a suitable angle, the airfoil deflects the oncoming air (for fixed-wing aircraft, a downward force), resulting in a force on the airfoil in the direction opposite to the deflection. Various airfoils serve different flight regimes. When an airfoil or any wing moves through air, the flow of air splits up and passes above and below the airfoil.

![Figure 1: Nomenclature of Airfoil](image)

Airfoil design is a major facet of aerodynamics. Airfoil’s upper surface is designed in such a way that the air rushing over the top of the airfoil, speeds up and stretches out. This decreases the air pressure above the airfoil. The air flowing below the airfoil moves in a comparatively straighter line, so its speed and air pressure remain the same. Since high air pressure always moves towards low air pressure, the air below the wing pushes upward towards the air above the wing. Bernoulli’s principle finds its application in this aspect.

Some basic terminologies associated with airfoils are stated below:

1. **The chord line**: It is the connection line between leading and trailing edge and it is straight.
2. **The chord length**: It is the length of the chord line.
3. **Angle of attack**: It is defined as the angle between the chord line and relative wind.
4. **The leading edge**: It is the front point of an airfoil where we get maximum curvature chord. It happens to be the length of the chord line.
5. **The trailing edge**: It is the part of an airfoil where we find maximum curvature.
6. **Total aerodynamic force (TAF)**: It is the total force on the airfoil produced by the airfoil shape and relative wind.
7. **Lift**: It is the force that directly opposes the weight of an airplane and holds the airplane in the air.
8. **Drag**: It is the aerodynamic force that opposes an aircraft's motion through the air.
9. **Camber**: It is the maximum distance between the mean camber line and the chord line, measured perpendicular to the chord.
10. **Maximum thickness**: It is the maximum separation from the bottom edge to the top edge. It is generally 0.12c or 12% of the chord.
11. **Turbulence or turbulent flow**: It is a flow regime in fluid dynamics characterized by chaotic changes in pressure and flow velocity. It is in contrast to a laminar flow regime.
12. **Critical angle of attack**: It is the angle of attack which produces maximum value of lift. It is also called stall angle of attack.

II. PROBLEM DEFINITION

In this work, the center of activity is to analyze the effect of flow performance of the airfoil, stall region
and calculation of its optimum angle of attack to obtain maximum value of lift to drag ratio which would result in better efficiency. A higher or more favorable L/D ratio is typically one of the major goals in aircraft design. The entire analyzation process involves monitoring of pressure distribution, velocity distribution, various aerodynamic forces, lift to drag ratio, stall region and critical angle of attack over the NACA 2412 airfoil at a high Reynolds number. A high Reynolds number, 7.6659E+06 is taken to conduct the experimentation through the shear stress transport turbulence model.

III. SST TURBULENCE MODEL

Turbulence modeling is the construction and use of a model to predict the effects of turbulence in the study of CFD. A turbulent fluid flow has features on many different length scales, which all interact with each other. A common approach is to average the governing equations of the flow, in order to focus on large-scale and non-fluctuating features of the flow. However, the effects of the small scales and fluctuating parts must be modelled. One of the most effective is the Shear Stress Transport (SST) model of Menter. The model combines the k-omega turbulence model and k-epsilon turbulence model such that the k-omega is used in the inner region of the boundary layer and switches to the k-epsilon in the free shear flow. The model works by solving a turbulence/frequency-based model (k-ω) at the wall and k-ε in the bulk flow. A blending function ensures a smooth transition between the two models. The formulation of the SST model is based on physical experiments and attempts to foretell solutions to typical engineering problems. Mathematical equations of the SST model are shown below

\[
\frac{\partial (\rho k)}{\partial t} + \frac{\partial (\rho u_i k)}{\partial x_j} = \rho P - \beta^* \rho_0 k + \frac{\partial}{\partial x_j} \left[ (\mu + \sigma_k \mu_t) \frac{\partial k}{\partial x_j} \right]
\]

\[
\frac{\partial (\rho \omega)}{\partial t} + \frac{\partial (\rho u_i \omega)}{\partial x_j} = \frac{\gamma}{\nu_t} \left[ (\mu + \sigma_\omega \mu_t) \frac{\partial \omega}{\partial x_j} \right] + 2(1 - F_1) \frac{\sigma_{\omega 2}}{\omega} \frac{\partial k}{\partial x_j} \frac{\partial \omega}{\partial x_j}
\]

Variable Definition:

\[ P = \tau_{ij} \frac{\partial u_i}{\partial x_j} \]

\[ \tau_{ij} = \mu_t \left( 2 s_{ij} - \frac{2}{3} \frac{\partial u_k}{\partial x_k} \delta_{ij} \right) - \frac{2}{3} \rho k \delta_{ij} \]

\[ S_{ij} = \frac{1}{2} \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \rho a_{ij} \]

\[ \mu_t = \max(\alpha_1 \omega - F_2, \max(\alpha_2 \omega, F_2)) \]

\[ \phi = F_1 \phi + (1 - F_1) \frac{\partial}{\partial x_i} \]

\[ F_1 = \tanh(\arctan(\sqrt{1 + \frac{\gamma}{\nu_t}} \sqrt{\frac{\phi}{\tanh(\arctan(\sqrt{1 + \frac{\gamma}{\nu_t}}))} \frac{\partial k}{\partial x_i}}} \right)^2) \]

\[ \frac{\partial k}{\partial x_i} \frac{\partial \omega}{\partial x_i} \]

\[ F_2 = \tanh(\sqrt{\frac{\gamma}{\nu_t} \frac{\partial k}{\partial x_i} \frac{\partial \omega}{\partial x_i}}} \right)^2) \]

\[ arg g_1 = \max \left[ \sqrt{k} - \frac{5000}{\sqrt{\omega}} \frac{\rho \omega^2 k}{C_{D \omega}} \right] \]

\[ arg g_2 = \max \left[ \frac{2\rho \omega^2}{\beta \omega'd^2} \frac{\partial k}{\partial x_i} \frac{\partial \omega}{\partial x_i} \right] \]

Constant:

\[ K – W Closure: \sigma_1 = 0.85, \sigma_2 = 0.65, \beta_1 = 0.075 \]

\[ K – e Closure: \sigma_2 = 1.00, \sigma_2 = 0.856, \beta_2 = 0.0828 \]

\[ SST Closure Constants: \beta^* = 0.09 \alpha_1 = 0.31 \]

Far Field Conditions:

\[ \frac{U_{\infty}}{L} < \omega_{farfield} < 10 \]

\[ \frac{10^{-5} U_{\infty}^2}{R_e} < k_{farfield} < 0.1 \frac{U_{\infty}^2}{R_e} \]

Boundary/Wall Conditions:

\[ \omega_{wall} = 10 \frac{U_{\infty}}{\beta_1 (\Delta r_i)^2} \]

IV. CFDMODELLING AND MESHING

4.1. Model Preparation-

Using the Geometry Modeler of Ansys Fluid Flow (CFX), the imported coordinates are processed to prepare the model. Contour is also generated to conduct the simulation.

4.2. Meshing-

Mesh generation is the practice of generating a polygonal or polyhedral mesh that approximates a geometric domain. Typical uses of meshing are for rendering to a computer screen or for physical
simulation such as finite element analysis (FEA) or computational fluid dynamics (CFD). For the analysis of the flow of fluid, the domains are needed to split into smaller sub domains and mesh accuracy of the domain increases as we go towards the airfoil shape. The governing equations are then discretized and solved inside each of these sub domains.

VI. RESULTS AND DISCUSSION

6.1. Pressure Plot-

Figure 5: Pressure contour at 2° AoA

Figure 6: Pressure contour at 7° AoA

6.2. Velocity Plot-

Figure 7: Velocity plot at 3° AoA

Figure 8: Velocity plot at 10° AoA

V. BOUNDARY CONDITIONS AND INPUTS

The most fundamental part of any CFD problem is the definition of its boundary conditions. The problem for the said airfoil model considers turbulent flow around the airfoil while maintaining variation in its angle of attack. To conduct the simulation, boundary conditions for the problem and some initial inputs are taken, and it is shown in the table below:

<table>
<thead>
<tr>
<th>Sl no</th>
<th>Input</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Analysis type</td>
<td>Steady state</td>
</tr>
<tr>
<td>2</td>
<td>Fluid Velocity</td>
<td>4.9928E+01</td>
</tr>
<tr>
<td>3</td>
<td>Mach Number</td>
<td>0.15</td>
</tr>
<tr>
<td>4</td>
<td>Operating Temperature</td>
<td>25° C</td>
</tr>
<tr>
<td>5</td>
<td>Operating Pressure</td>
<td>172252 Pa</td>
</tr>
<tr>
<td>6</td>
<td>Global Length</td>
<td>1.3974E+00</td>
</tr>
<tr>
<td>7</td>
<td>Fluid Density</td>
<td>2.0118E+00</td>
</tr>
<tr>
<td>8</td>
<td>Turbulence Model</td>
<td>SST</td>
</tr>
<tr>
<td>9</td>
<td>Reynolds Number</td>
<td>7.6659E+06</td>
</tr>
<tr>
<td>10</td>
<td>Fluid Viscosity</td>
<td>1.8310E-05</td>
</tr>
</tbody>
</table>

Table 1: Various inputs and values
6.3. Total Aerodynamics Force Plot

The aerodynamic force depends on the square of the velocity. Doubling the velocity quadruples the force. The dependence of lift and drag on the square of the velocity has been known for more than a hundred years. The Wright brothers used this information in the design of their first aircraft.

![Figure 9: Total Aerodynamics Force at 1º AoA](image)

The L/D chart shows that the value of lift to drag ratio reaches maximum at 5º angle of attack. The higher the lift to drag ratio, better the efficiency.

6.5. Lift and Drag Forces

The lift to drag ratio is an important parameter in measuring the airfoil performance. When the Lift to Drag ratio is highest at a particular angle, it means that the drag force has less value and that particular angle provides greater efficiency which directly leads to better fuel economy in aircraft and climb performance.

The values of Lift Force and Drag Force and their ratios are shown below:

<table>
<thead>
<tr>
<th>AoA</th>
<th>Lift [N]</th>
<th>Drag [N]</th>
<th>Ratio</th>
</tr>
</thead>
<tbody>
<tr>
<td>0º</td>
<td>5.54046</td>
<td>0.245552</td>
<td>22.56329</td>
</tr>
<tr>
<td>1º</td>
<td>8.22491</td>
<td>0.264518</td>
<td>31.09395</td>
</tr>
<tr>
<td>2º</td>
<td>10.8805</td>
<td>0.292893</td>
<td>37.14838</td>
</tr>
<tr>
<td>3º</td>
<td>13.5195</td>
<td>0.328174</td>
<td>41.19613</td>
</tr>
<tr>
<td>4º</td>
<td>16.167</td>
<td>0.374772</td>
<td>43.13823</td>
</tr>
<tr>
<td>5º</td>
<td>18.7896</td>
<td>0.430188</td>
<td>43.67765</td>
</tr>
</tbody>
</table>

In general, the greater the angle of attack, the more lift is generated by the airfoil. However, this is only true to a point. At some point, the airfoil reaches its critical or stall angle of attack. Beyond the critical angle of attack, the stall region, where the airfoil

![Figure 10: Total Aerodynamics Force at 5º AoA](image)

![Figure 11: Lift to Drag Ratio](image)

![Figure 12: Critical Angle of Attack and Stall](image)

Table 3: Values of Lift & Drag at various angles

<table>
<thead>
<tr>
<th>AoA</th>
<th>Lift [N]</th>
<th>Drag [N]</th>
</tr>
</thead>
<tbody>
<tr>
<td>12º</td>
<td>34.8231</td>
<td>1.07316</td>
</tr>
<tr>
<td>14º</td>
<td>37.4971</td>
<td>1.38883</td>
</tr>
<tr>
<td>15º</td>
<td>39.1772</td>
<td>1.526</td>
</tr>
<tr>
<td>16º</td>
<td>39.1694</td>
<td>1.76623</td>
</tr>
<tr>
<td>18º</td>
<td>38.5277</td>
<td>2.37823</td>
</tr>
<tr>
<td>21º</td>
<td>30.2649</td>
<td>4.52857</td>
</tr>
<tr>
<td>25º</td>
<td>25.5317</td>
<td>7.30268</td>
</tr>
</tbody>
</table>
is no longer effectively generating lift and the airplane is stalled. In this stall region, the amount of drag generated by the airfoil increases dramatically as the lift decreases.

**CONCLUSIONS**

This research work presents the simulated flow over a NACA 2412 airfoil model with a high Reynolds number at various angles of attack.

1. In all cases, it has been observed that the lift force and drag force increase respectively as the angle of attack increases until it reaches stall.

2. 5° angle of attack has been found to be the optimum angle. At this specific angle, the said airfoil model produces the maximum ratio of Lift & Drag (43.67765), which is ideal for efficient performance compared to any other angles of attack.

3. For an angle of attack of 5°, it was observed that the maximum pressure reaches 2.561e+003 (Pa) at its bottom, and at top it experiences a minimum pressure of -3.164e+003, and maximum total aerodynamics force stands at 8.456e-002 (N)

4. It has been found that airfoil experiences low pressure at its top surface, whereas high pressure at its bottom. This occurs because when a plane moves forward, air pushes on the bottom of the airfoil resulting in an increase in pressure. Meanwhile, air moves faster over the top of a wing, which results in an area of lower pressure. The difference between the high and low pressure generates lift. This phenomenon justifies Bernoulli’s Principle which states that pressure decreases when air moves faster.

5. 15° angle has been found to be the Critical angle of attack or stall angle of attack. Beyond this particular angle, the amount of lift generated by the airfoil drops dramatically and this particular region is called stall. This occurs because of the flow of air which gets separated from the airfoil, forming turbulent eddies that destroy the ability of the airfoil to generate lift.

**ACKNOWLEDGMENTS**

I would like to express my sincere gratitude to my guide, Assistant Professor, Shaheen Beg Mughal for providing me the opportunity to work with him and his continuous assistance throughout the course of this research work. At last, I thank my parents who gave me an upbringing to carry on this work.

**REFERENCES**