SPALART ALLMARAS UNSTEADY FLOW INVESTIGATION
USING COMPUTATIONAL FLUID DYNAMICS FOR DRAG AND LIFT
FORCE OF AN AIRFOIL

Nitesh Verma¹, Surya Prakash Chauhan², Kundan Singh Deshwal³
nitesverma1305@gmail.com, csurya97@gmail.com, kundandeshwal88@gmail.com
Department of Mechanical Engineering
Seemant Institute of Technology, Pithoragarh

Abstract

The studies of drag and lift are subjects of importance in fluid dynamics for the engineering benefits that they can offer are highly significant. This paper presents SpalartAllmaras flow investigation. In this paper, a computational fluid dynamics CFD model is studied for unsteady external flow. Drag and lift experienced by a solid object in a flowing fluid are the resultant resistance forces established by the components of the resultant aerodynamic force along and normal to the free stream velocity respectively. In automotive design, good aerodynamic consideration aims for the least drag to achieve efficiency, and also to optimize negative lift particularly in motor sport. A proposed alternative to this problem would be the use of Computational Fluid Dynamics (CFD) modeling to obtain the required output data for the calculations of drag and lift properties. In this work ANSYS FLUENT 12.0.16 software was employed to perform the task. Before this method can be used to simulate the drag and lift for a practical design object, a test on its capability must be carried out. This was done through the simulation of fluid flow over a simple model of NACA 4412 Airfoil. The pressure distribution and the corresponding spatial co-ordinates were then extracted into a spreadsheet file in order to calculate the coefficients of drag and lift. The results from the simulation were found to correlate well with the ones obtained from the experiment.

Problem Specification and boundary condition

The purpose of analysis is to measure the aerodynamic forces and total pressure at the tip of NACA 4412 Airfoil and to determine the lift and drag coefficient at a 2° angle of attack. In this study work present result based on the experimental data of “Theory of Wing Sections” By Ira Herbert Abbott, Albert Edward Von Doenhoff, At the Temperature 288.17 K. air flow over the external surface of airfoil. The cord of Airfoil is 100 mm and Mach no 0.15

<table>
<thead>
<tr>
<th>Properties of air at 288.17 K</th>
<th>Dimension parameters of Airfoil</th>
</tr>
</thead>
<tbody>
<tr>
<td>Density: 1.2250 Kg/m³</td>
<td>Cord: 100 mm</td>
</tr>
<tr>
<td>Kinematics Viscosity: 1.4607 × 10⁻⁵</td>
<td>Area: 100 m²</td>
</tr>
<tr>
<td>Specific heat: 1.4 kj/kg K</td>
<td>Mack No: 0.15</td>
</tr>
<tr>
<td>-</td>
<td>Length of trailing edge: 0.02c</td>
</tr>
<tr>
<td>-</td>
<td>Angle of attack: 2°</td>
</tr>
</tbody>
</table>

Table 1: Boundary condition of the specific problem

Analytical formulation

The pressure coefficient can be expressed by

\[
C_p = \frac{p - p_\infty}{\frac{1}{2} \rho U_0^2}
\]

Lift coefficient

The lift of the airfoil can also be expressed as
\[
\text{Lift} = F_L = \iint_{\text{surface}} dF_L = \left[ \int_0^L dF \cos \theta \right]_{\text{top}} + \left[ \int_0^L dF \cos \theta \right]_{\text{bottom}}
\]

For simplicity this span is considered uniform, because of this, integration is only necessary over L given

\[
\int_0^L dF \cos \theta = \int_0^L ps \, dl \cos \theta = \int_0^L (p - p_\infty) s \, dl \cos \theta
\]

It is known that \( \cos \theta = \frac{dx}{dl} \) yielding,

\[
\int_0^L (p - p_\infty) s \, dl \frac{dx}{dl} = s \int_0^L (p - p_\infty) \, dx
\]

With the distance along the chord defined as \( dx = dc \cos \alpha \),

\[
S \int_0^L (p - p_\infty) \cos \alpha \, dc = F_L
\]

\[
C_L = \frac{F_L}{\frac{1}{2} \rho U_\infty^2 A}
\]

The overall drag coefficient defined in the usual manner is

\[
C_D = \frac{F_D}{\frac{1}{2} \rho U_\infty^2 A}
\]

Where A is the area of the object.

Result

Computational Fluid Dynamics (CFD) has shown to be adequate for predicting the pressure forces on Airfoil. Over the years extensive work has been done in its development and improvements in its ability to predict. But still a comparison between numerical simulations and experiments shows differences, which can be due to inadequate numerical modeling or measurement errors in the experiments. In practice today, the CFD results for a certain case are compared to experimental results and then, if found good, the numerical results of other similar cases are considered as accepted.

The coefficient of Drag and coefficient of Lift for regions were also compared with experimental results and are shown in Table 4-2. At the surface, due to velocity, the difference in numerical and experimental values is high. Though coefficient of forces at the wall surface was different, the area averaged values are not that different.

<table>
<thead>
<tr>
<th>Coefficient of forces</th>
<th>Experiment</th>
<th>Numerical</th>
</tr>
</thead>
<tbody>
<tr>
<td>( C_L )</td>
<td>0.649</td>
<td>0.654</td>
</tr>
<tr>
<td>( C_D )</td>
<td>0.007</td>
<td>0.00765</td>
</tr>
</tbody>
</table>

Table 2: difference between experimental and numerical values of drag and lift forces

Figure 1: comparison of limit coefficient in Experimental and numerical value
D. Anderson and S. Eber-hardt, “How Airplanes Fly: A Physical Description of Lift,” the Aviation History Online Museum,

David M. Smiadak “Lift Coefficient for an Airfoil”

“UIUC Airfoil Coordinates Database,” http://amber.aae.uiuc.edu/~mselig/ads/coord_database.html


