Numerical analysis of two-dimensional wing with trailing edge flap

Mrs. Surekha Rathi Samundi D., Shakti Uma Devi S., Pracheesh P., and Thirumalai Kumar M.,
Mail id: shakthiumadevi@gmail.com

Abstract—This paper investigates the critical performance parameters of the flow over an airfoil with trailing edge flaps when they are deflected, at different angles of attack computationally using three turbulence models namely the Spalart-Allmaras model, \( k-\varepsilon \) model, \( k-\omega \) model. Further, the results obtained are compared with each other. Based on the results obtained, the appropriate turbulence model for analysis of flow over a wing can be determined.

Keywords – Airfoil analysis, Turbulence model, Flap, Computation

I. INTRODUCTION

The computational method has become widely popular in the recent days because of its cost-effectiveness and less time consumption. But several problems take much time to be computed based on the viscous model used for flow analysis. Hence, there has always been a need to justify the appropriate usage of turbulence model based on the nature of the problem. Here, a normal airfoil analysis is carried out to explore the variation in results using different viscous models. The airfoil analysis can be carried out using ANSYS Fluent for various trailing edge flap deflection. When flaps are deflected, it causes turbulence and separated flow; hence it becomes very essential to choose a perfect solution model in order to get accurate results. The computation is carried out at lower Reynolds number in an incompressible subsonic flow. The airfoil is fitted with flap at 20% of chord from the trailing edge. The reference aircraft which is taken for the analysis is the HAL-Kiran aircraft which is the in-house aircraft created by the Hindustan Aeronautics Limited.

In the present study, the airfoil with trailing edge flap is analyzed using three turbulence models respectively – Spalart Allmaras model, \( k-\varepsilon \) model and \( k-\omega \) model.

II. THEORETICAL FORMULATION

The Computational fluid dynamics software available in ANSYS is the Ansys Fluent which uses various cfd solvers to solve the basic governing equations namely the continuity, momentum and the energy equation. Since we are going to carry out the computation in the low-speed subsonic flow, the flow is considered to be incompressible, so the density may be considered as constant whereas the pressure is considered a variable. The governing equations are given for the two-dimensional case since we are going to compute the flow around the two-dimensional wing only.

\[
\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \mathbf{u}) = \frac{\partial \rho \mathbf{u}}{\partial t} = \frac{\rho \mathbf{u} \cdot \nabla \mathbf{u}}{\text{constant}} + \rho f (\mathbf{u}, \nabla \mathbf{u})
\]

\[
\rho \left[ \frac{\partial \mathbf{u}}{\partial t} + \mathbf{u} \cdot \nabla \mathbf{u} \right] = -\nabla p + \nabla \cdot \tau + \mathbf{f}
\]

\[
\frac{\partial u}{\partial t} + \mathbf{u} \cdot \nabla u = -\frac{\nabla p}{\rho} + \frac{1}{\rho} \left[ \nabla \cdot \left( \nabla \frac{p}{\partial t} \right) \right] + \frac{1}{\rho} \left[ \nabla \cdot \left( \nabla \frac{p}{\partial t} \right) \right] - \frac{1}{\rho} \left[ \nabla \cdot \left( \nabla \frac{p}{\partial t} \right) \right] - \frac{1}{\rho} \left[ \nabla \cdot \left( \nabla \frac{p}{\partial t} \right) \right]
\]

\[
\rho \left[ \frac{\partial \mathbf{u}}{\partial t} + \mathbf{u} \cdot \nabla \mathbf{u} \right] = -\nabla p + \nabla \cdot \tau + \mathbf{f}
\]

\[
\rho \left[ \frac{\partial \mathbf{u}}{\partial t} + \mathbf{u} \cdot \nabla \mathbf{u} \right] = -\nabla p + \nabla \cdot \tau + \mathbf{f}
\]

\[
\rho \left[ \frac{\partial \mathbf{u}}{\partial t} + \mathbf{u} \cdot \nabla \mathbf{u} \right] = -\nabla p + \nabla \cdot \tau + \mathbf{f}
\]

\[
\rho \left[ \frac{\partial \mathbf{u}}{\partial t} + \mathbf{u} \cdot \nabla \mathbf{u} \right] = -\nabla p + \nabla \cdot \tau + \mathbf{f}
\]

\[
\rho \left[ \frac{\partial \mathbf{u}}{\partial t} + \mathbf{u} \cdot \nabla \mathbf{u} \right] = -\nabla p + \nabla \cdot \tau + \mathbf{f}
\]
B. K-ε model

The K-ε model is the two-equation turbulence model which solves two separate transport equations to find the turbulent velocity and the length scales to be independently determined. This model was proposed by Launder and Spalding in the year 1974 and been used to solve many practical engineering flow problem where turbulence plays a dominant role.

The modelled transport equation is given by

\[
\begin{align*}
  \frac{\partial D}{\partial t} &= \nabla \cdot \left( \frac{\rho k}{\varepsilon} \nabla \right) + \beta^\prime \rho \omega k + \frac{\partial}{\partial x_j} \left[ \mu_\varepsilon (S_{ij} - \varepsilon \delta_{ij}) \right] \\
  \frac{\partial \rho \omega}{\partial t} &= \nabla \cdot \left( \frac{\rho \omega^2}{\mu_\omega} \nabla \right) + \frac{\partial}{\partial x_j} \left[ \mu_\omega (S_{ij} - \omega \delta_{ij}) \right] + 2\rho (1 - F_1) \frac{1}{\omega} \frac{\partial k}{\partial x_j} \frac{\partial \omega}{\partial x_j}
\end{align*}
\]

Where,

\[
\begin{align*}
  \beta &= F_4 \beta_2 + (1 - F_2) \beta_2, \\
  \gamma &= F_1 \gamma_1 + (1 - F_2) \gamma_2, \\
  \sigma &= F_1 \sigma_1 + (1 - F_2) \sigma_2, \\
  \sigma_c &= F_1 \sigma_c + (1 - F_2) \sigma_c
\end{align*}
\]

\[
F_2 = \tanh \left[ \max \left( \frac{2 \sqrt{k}}{0.09 \omega}, \frac{5000 \nu}{\gamma^2 \omega} \right) \right]^2
\]

In the above equation, \( G_k \) represents the turbulent kinetic energy created by the mean velocity gradients and \( G_b \) represents the turbulence kinetic energy created by the buoyancy effects. \( M \) represents the fluctuating dilation in the turbulent flow which is compressible.

\[
C_1 = \max \left( \frac{n}{n + 5} \right), \quad n = \frac{k}{\varepsilon}, \quad S = \sqrt{2 \varepsilon U_L S_u}
\]

C. k-ω model

The k-ω model is considered to be robust and has been considered to give accurate results to a wide variety of flows ranging from the flow over an airfoil, adverse pressure gradient flow and shock waves. The k-ω turbulence model is given by the following equations

\[
\begin{align*}
  f_{r_1} &= C_{r_1} \exp \left[ -C_{r_2} (\nu / \nu)^2 \right] (11) \\
  f_{r_2} &= C_{r_3} \exp \left[ -C_{r_4} (\nu / \nu)^2 \right] (10)
\end{align*}
\]

III. COMPUTATIONAL ANALYSIS OF THE AIRFOIL

The computational analysis of the airfoil is carried using the Ansys fluent software. Computational analysis is going to be carried out on the airfoil using the computation and design software namely Ansys Fluent and CATIA respectively. The ultimate aim of the analysis is to compare the results such as the lift coefficient, drag coefficient, contour plots obtained by solving using three solver models.
Choosing a perfect and appropriate solver model goes a long way in getting accurate results with less server time. It requires a lot of experience in the field to choose a right model because there is no perfect model for the particular problem. Before carrying out the experimental analysis on the wing, it is always mandatory to find the forces acting on the airfoil experimentally. The computational results can be compared with the experimental results and then the solver model which gives the nearest solution can be used for the analysis of the wing computationally.

The following steps are involved in successfully carrying out a computational analysis. They are
A. Design of the geometry
B. Defining the fluid domain
C. Meshing
D. Setup
E. Solution

A. DESIGN OF THE GEOMETRY

The design of the airfoil requires coordinates which were imported from the airfoil tools available online. The aircraft which is taken for analysis is the HAL KIRAN AIRCRAFT, which is made up of NACA 23015 at the root and NACA 23012 at the tip. In order to carry out the two-dimensional analysis of the wing, the airfoil must be the average of the root airfoil and the tip airfoil. This linear regression morphing is used to obtain the average airfoil. The coordinates of the root and tip are interpolated and the airfoil coordinates are plotted using the open source application called the profscan which has the capability to transform the day file onto the dxf file which is compatible with AutoCAD as well as CATIA. This dxf file is imported in CATIA which will be around 100 m which must be scaled to the size of our model to be fabricated. The scale ratio is given as 1.67 which scales up the model dimension to 167 mm. Then an extended surface is created where we need to fix our trailing edge flap. Then the front part and the rear part of the airfoil is split thus creating two separate surfaces. Then a hinge is created at a distance of about 20% of the chord length from the trailing edge. Rotation operation is used to create deflected flap. For various flap selection angles, the resultant geometry is saved in the stp file format which is compatible with ansys software. The stp file of the airfoil is imported into the design modeller of the Fluent.

B. DEFINING THE FLUID DOMAIN AND THE AIRFOIL

The fluid domain is created around the airfoil is the C-domain which extends about 5 times the chord length from the leading edge and about 10 times the chord from the trailing edge. The domain is created using the sketch and surface is created using the Add frozen option. Then the boolean operation is used to subtract the airfoil from the domain. After this, an internal zone in the shape of a circle is created with its diameter about five times the chord length of the airfoil. Finally, projection tool is used to separate the internal zone and the external zone.
C. MESHING

After completing the design in the design modeller, the meshing option is double clicked then the sizing option is chosen. The proximity and curvature option is selected from the advanced size function. The smoothing should be made high with fine relevance. The internal zone should have an element size of 0.5 cm and the external zone should have an element size of 1 cm.

Table 1: Number elements for the fluid domain with various flap deflection angles

<table>
<thead>
<tr>
<th>S.No</th>
<th>Flap deflection angles in degrees</th>
<th>Number of elements</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>0</td>
<td>87,000</td>
</tr>
<tr>
<td>2</td>
<td>10</td>
<td>90,000</td>
</tr>
<tr>
<td>3</td>
<td>20</td>
<td>1,17,000</td>
</tr>
<tr>
<td>4</td>
<td>30</td>
<td>1,54,000</td>
</tr>
</tbody>
</table>

From the above table we could infer that as the flap deflection angle increases for the airfoil, the corresponding number of elements also increases so that the results obtained are good.

D. SETUP

The meshing is completed and after that, the edges and the faces are named so that the boundary conditions can be given during the setup phase of the computation. The semi-circle upstream of the airfoil is named inlet and the line downstream of the airfoil is named as the outlet. The edges below and above are named symmetry. The airfoil shape is named as the airfoil surface. After naming all the edges, the faces are also named. The face of the inner circle is named as the internal zone and the face of the external zone is named as the external zone.

The parallel solver is chosen, so that computation can be solved quickly. The model can be chosen as spalart Allmaras, K-ε, k-ω model each time and the results must be obtained. The air with standard properties is chosen as the material for the fluid domain. The boundary conditions are
given at the inlet as the velocity inlet and the pressure outlet.

The computational analysis on the airfoil is going to be carried out for various flap deflection ($\delta_f$) angles namely 0, 10, 20, 30 degrees and in addition to that at a particular deflection angle solution are going to be obtained at various angle of attacks using the three models mentioned above.

In order to change the angle of attack, the velocity components given at the inlet during the boundary condition s are only changed, other than that all the steps are same till the meshing. So as a whole 36 computations are carried out.

The various inputs given in the setup stage are given in the table 2 given below

Table 2: Conditions given in the setup stage

<table>
<thead>
<tr>
<th>S.No</th>
<th>Conditions</th>
<th>Input</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Model</td>
<td>Spalart Allmaras model</td>
</tr>
<tr>
<td></td>
<td></td>
<td>k-epsilon model</td>
</tr>
<tr>
<td></td>
<td></td>
<td>k-omega model</td>
</tr>
<tr>
<td>2</td>
<td>Material</td>
<td>Air</td>
</tr>
<tr>
<td>3</td>
<td>Boundary conditions</td>
<td>Inlet-Velocity inlet Outlet-Pressure outlet</td>
</tr>
<tr>
<td>4</td>
<td>Monitors</td>
<td>Lift and drag on the airfoil surface</td>
</tr>
<tr>
<td>5</td>
<td>Initialization</td>
<td>Hybrid Initialization</td>
</tr>
</tbody>
</table>

We are going to compute the forces acting on the airfoil at various angles of attack at a constant flap deflection angle. Instead of changing the orientation of the geometry, the flow direction is altered. The flow is resolved into the horizontal and vertical components such that it produces the same effect as changing the angle of attack. Thus, the velocity-inlet boundary conditions vary for each and every angle of attack. The velocity inlet is given in terms of the x and y velocity components.

Table 3: Components of velocity at various angles of attack

<table>
<thead>
<tr>
<th>S.no</th>
<th>Angle of attack</th>
<th>$V_x$ (m/s)</th>
<th>$V_y$ (m/s)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>0</td>
<td>40</td>
<td>0</td>
</tr>
<tr>
<td>2</td>
<td>5</td>
<td>39.847788</td>
<td>3.486230</td>
</tr>
<tr>
<td>3</td>
<td>10</td>
<td>39.392310</td>
<td>6.943927</td>
</tr>
</tbody>
</table>

Thus the conditions are given and the solution is calculated after the initialization till the convergence is achieved.

E. SOLUTION

After the convergence of the solution, the pressure contour, velocity contour, Streamlines of the flow, lift and the drag coefficients are obtained. The results are then tabulated and compared.

IV. RESULTS AND DISCUSSION

This part constitutes the main part because all the computational results are tabulated and the comparative study is carried out. First, the lift and drag forces obtained using various models are compared at various angle of attack and followed by which the contours and XY plots are analyzed.

Table 4: Coefficient of Lift and Drag

<table>
<thead>
<tr>
<th>$\delta_f$</th>
<th>Spalart-Allmaras</th>
<th>$K-e$</th>
<th>$K-\omega$</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>$C_l$</td>
<td>$C_d$</td>
<td>$C_l$</td>
</tr>
<tr>
<td>0</td>
<td>0.043</td>
<td>0.016</td>
<td>0.002</td>
</tr>
<tr>
<td>10</td>
<td>0.572</td>
<td>0.039</td>
<td>0.048</td>
</tr>
<tr>
<td>20</td>
<td>0.691</td>
<td>0.040</td>
<td>0.047</td>
</tr>
<tr>
<td>30</td>
<td>0.744</td>
<td>0.050</td>
<td>0.09</td>
</tr>
</tbody>
</table>

Figure 9: Comparison of Coefficient of lift of the turbulence models
The above graph is plotted between the flap deflection angles and the coefficient of lift and for various models for the same angle of attack of 0 degrees. Furthermore, we could infer that as the flap deflection angle increases the coefficient of lift also increases. The pattern in which the coefficient of lift increases for various models are the same but the values obtained are different for different models. Shown below in Figure are the pressure and velocity contours of the Airfoil when they are deflected to 20 degrees at the constant angle of attack 5 degrees using the 3 models.

**Figure 9: Comparison of Coefficient of drag of the turbulence models**

a. Spalart Allmaras model

b. k-Epsilon model

c. K-omega model

**Figure 10 Pressure contours**

a. Spalart Allmaras model

b. K-epsilon model

c. K-omega model

**Figure 11: Velocity contours**

a. Spalart Allmaras model

b. K-epsilon model

c. K-omega model
It can be observed that the contour plots more or less yields the same results irrespective of the models. Only by carrying out the experiments the appropriate model could be found. But we could infer from the table and the graphs that when the flaps are used the increment in the lift is about 0.7 which is the desirable range for the plain flap.

V. CONCLUSION

Thus, based on the computational analysis of airfoil carried out at various angles of attack and flap deflection angles using the three turbulence models, we could conclude the results obtained varies by changing the model. In order to get accurate results, the perfect model must be chosen. After carrying out test on the airfoil with same dimensions in the wind tunnel, the appropriate model can be chosen and used for the computation in the wing computation. From the experimental results available in the Theory of Wing sections, the results obtained by using Spalart Allmaras model were in good agreement with experimental results.

ACKNOWLEDGEMENT

We would like to thank Mr. Madhu D., Manager of Structural Hawk Assembly, Hindustan Aeronautics Limited for sharing with us the real-time application of flaps and also for guiding us continuously for the completion of the project.

REFERENCES
