Numerical Simulation of Turbulent Flow around Hydrofoil by Spalart-Allmaras Model

Md. Shahjada Tarafder¹, Md. Syful Isalm Bipul²

¹Department of Naval Architecture and Marine Engineering, Bangladesh University of Engineering and Technology Dhaka-1000, Bangladesh
E-mail: shahjada68@yahoo.com

²Department of Naval Architecture & Offshore Engineering, Bangbandhu Sheikh Mujibur Rahman Maritime University, Bangladesh, Mirpur-12, Dhaka-1216
E-mail: naoe17011.bipul@bsmrmu.edu.bd

ABSTRACT

The objective of this research paper is to use COMSOL MULTIPHYSICS software to simulate turbulent flow around two-dimensional hydrofoil by Spalart-Allmaras model. The computational results for the hydrodynamic performance of S809 hydrofoil at various angles of attack operating at Reynolds numbers of Re = 2 × 10⁹ were compared to existing experimental data from reliable sources using S809 hydrofoil. Notably, an unstructured mesh with 1,11,084 cells had already been identified as the best fit for numerical simulations. The impact of various turbulence models, especially Spalart–Allmaras on the predicted hydrodynamic forces like lift, drag, pressure are also analyzed. The computed result showed that the coefficient of lift, drag and pressure increased as angle of attack increased. From pressure contours, it was noticed that there was no flow separation at low angle of attacks but began to separate at high angle of attacks. Velocity contours and streamlines are also analyzed.

Key words: CFD, NREL, S809, hydrofoil, Spalart Allmaras model.

1. INTRODUCTION

Hydrofoils are lifting surfaces that have been used in the marine industry for many decades including sailing, defense, and passenger craft. There are numerous available shapes of foils (L-foils, C-foils, vertical foils) that are attached to the hull of a vessel in different configurations. As the forward speed of the craft increases, they induced a lift force that makes the hull rise out of the water. In this way wave and skin friction drag are reduced, allowing for significantly higher speeds with the same amount of power supplied. Some interesting work on CFD analyses of the flow around the S809 are presented below: Wolfe and Ochs [1] analyzed the aerodynamic characteristics of S809 airfoil for laminar flow in steady-state condition. Gupta and Gordon Leishman [2] modified the Leishman-Beddoes dynamic stall model to represent the unsteady aerodynamic behavior of the S809 airfoil for wind turbine applications. According to Qu et al. [3], the Reynolds number has a direct impact on the aerodynamic performance of a wind turbine airfoil. In a dry environment, Douvi and Margaris [4] computed the S809 aerodynamic performance at various angles of attack while operating at Reynolds numbers. The S809 airfoil was developed by Dan M. Somers [5] who also computed the experimental findings. It has proven able to achieve the two main goals of low-profile drag and limited maximum lift.

A 2D Navier-Stokes simulation of steady and unsteady flow for the S809 airfoil was studied by Guerri et al. [6]. They compared between SST k-ω and RNG k-ε turbulence models. They demonstrated that the k-ω model performs better than the k-ε model during simulating the flow instability region. David Hartwanger et al. [7] investigated a CFD analysis for S809 airfoil and came to the conclusion that using a high-resolution mesh with advanced turbulence models provide an excellent match with experimental. Ariful, Karim and Rahman [8] analyzed water wave characteristics around submerged NACA 4412 hydrofoil and established amplitude of the wave, lift coefficients and drag coefficients values decreases with the increase in submergence depth. By employing the finite volume technique and a non-orthogonal body fitted grid, Tarafder and Mursaline [9] were able to simulate the turbulent flow around two-dimensional bodies. The problem was solved using the SIMPLE algorithm and a simplified pressure correction equation for a collocated arrangement of scalar and vector variables.

This paper is concerned with the flow around S809 at different angle of attack on 2D hydrofoil. For the study, the commercial computational fluid dynamic (CFD) code COMSOL MULTIPHYSICS based on finite volume approach is employed. An unstructured mesh is built and executed in COMSOL for a numerical iterative solution. At various angles of attack, the Spalart-Allmaras turbulence model has been used to
simulate turbulent flow past the hydrofoil. The computational results are compared to experimental data to validate them. The calculated results and experimental data have a good level of agreement. Finally, for various x/C ratios, the hydrofoil’s lift, drag, pressure contours, velocity magnitude contours, streamlines, and wall resolution are calculated at α = 0-14°, Re = 2 × 10^6.

**Nomenclature**

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Definition</th>
</tr>
</thead>
<tbody>
<tr>
<td>C</td>
<td>chord length</td>
</tr>
<tr>
<td>C_L</td>
<td>lift coefficient</td>
</tr>
<tr>
<td>C_D</td>
<td>drag coefficient</td>
</tr>
<tr>
<td>C_p</td>
<td>pressure coefficient</td>
</tr>
<tr>
<td>α</td>
<td>angle of attack</td>
</tr>
<tr>
<td>Re</td>
<td>Reynolds number</td>
</tr>
</tbody>
</table>

2. GOVERNING EQUATION

2.1 Continuity equation

\[ \frac{\partial u}{\partial t} + \frac{\partial (uv)}{\partial x} + \frac{\partial (vu)}{\partial y} = 0 \]  

(1)

The momentum equation in the direction for incompressible flows may be written as:

\[ \frac{\partial u}{\partial t} + \frac{\partial (uu)}{\partial x} + \frac{\partial (uv)}{\partial y} = \frac{1}{\rho} \frac{\partial P}{\partial x} \]  

\[ + \frac{\partial}{\partial y} \left[ (\bar{v} + \bar{v}_T) \frac{\partial u}{\partial y} \right] \]  

(2)

\[ \frac{\partial v}{\partial t} + \frac{\partial (vu)}{\partial x} + \frac{\partial (vv)}{\partial y} = \frac{1}{\rho} \frac{\partial P}{\partial y} \]  

\[ + \frac{\partial}{\partial x} \left[ (\bar{v} + \bar{v}_T) \frac{\partial v}{\partial x} \right] \]  

(3)

Using the rules of partial differentiation, invoking continuity equation for incompressible flow the second term becomes zero and using the definition of kinematic viscosity \( \frac{\mu}{\rho} = \nu \) we finally get

\[ \frac{\partial u}{\partial t} + \frac{\partial (uu)}{\partial x} + \frac{\partial (uv)}{\partial y} = -\frac{1}{\rho} \frac{\partial P}{\partial x} \]  

\[ + \nu \left( \frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2} \right) \]  

(4)

Similarly, this form of momentum equation in the y direction is given as:

\[ \frac{\partial v}{\partial t} + \frac{\partial (vu)}{\partial x} + \frac{\partial (vv)}{\partial y} = -\frac{1}{\rho} \frac{\partial P}{\partial y} \]  

\[ + \nu \left( \frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2} \right) \]  

(5)

2.2 Spalart Allmaras model:

The Spalart-Allmaras model is a one-equation model that solves the kinematic eddy viscosity transport equation (without calculating the length scale of the shear layer thickness). The Standard Spalart Allmaras model is one-equation turbulence model available to calculate the turbulence in unsteady state flow model (Houzeaux, 2002). This model involves an eddy-viscosity variable \( \nu \), related to the eddy-viscosity \( \nu_e \) by:

\[ \nu_e = f_{\nu_1} \nu \]  

(6)

Where:

\[ f_{\nu_1} = \frac{x^3}{x^3 + \frac{\nu}{\nu_e}} \]  

\[ x = \frac{\nu}{\nu_e} \]  

The transport equation for \( \nu \) is:

\[ \frac{\partial \nu}{\partial t} + U_j \nabla \nu = C_{nu} S \nu + \frac{1}{\sigma} \frac{\partial}{\partial x_j} \left[ \nu \frac{\partial \nu}{\partial x} \right] \]  

\[ + C_{nu2} \left( \frac{\partial \nu}{\partial x_j} \right)^2 - C_{nu1} f_{\nu_1} \frac{\nu^2}{d^2} \]  

(7)

where \( d \) is the short distance to the wall and \( k \) is the Von-Karman constant. The constants of the model are given in the set of constant below, where the function \( f_{\nu_1} \) is given by:

\[ f_{\nu_1} = g \left[ \frac{1 + C_{nu}^6}{g^6 + C_{nu}^6} \right] \]  

(8)

With:

\[ g = r + C_{\nu} \left( r^6 - r \right) ; r = \frac{\nu}{Sk^2 d^2} \]  

\( g \) can take relatively high values, so it is preferable to compute \( f_{\nu_1} \) as:

\[ f_{\nu_1} = \left[ \frac{1 + C_{nu}^3}{1 + C_{nu}^3} \right]^{rac{1}{6}} \]  

(9)

the production term involves the quantity \( S \)
which is a function of the magnitude of vorticity $S$ and is given by:
\[
S = S + \frac{\nu}{k^2} f_{v_2}; f_{v_2} = 1 - \frac{x}{1 + x f_{v_1}}
\]  
(10)

Where:
\[
S = \sqrt{2\Omega(u_j)\Omega(u_j); \Omega(u_j)} = \frac{1}{2} \left( \frac{\partial u_i}{x_j} - \frac{\partial u_j}{x_i} \right)
\]

and finally the constant are:
\[
C_{b1} = 0.1335; C_{b2} = 0.622; \sigma = 0.667; \\
C_{v1} = 7.1; k = 0.41; C_{aol} = \frac{C_{b1}}{k^2} + \frac{1 + C_{b2}}{\sigma}; \\
C_{a2} = 0.3; C_{a3} = 2.0
\]

The Standard Spalart Allmaras turbulence model can also be written as

The initial condition for $\nu$ is specified to be zero up to be zero up to a value of $\nu_x / 10$, that is, $\nu = 0 \rightarrow \nu_x / 10$. The boundary conditions are set according to:
1) At the inflow, $\nu = \nu_x$, 
2) At the solid surface, $\nu = 0$ 
3) At the outflow, extrapolation is used.

3. COMPUTATIONAL DOMAIN

A two-dimensional representation of the flow field around the hydrofoil was being used. Since the experiment was also conducted with a hydrofoil of the same length, the flow of an incompressible fluid from left to right using a hydrofoil with a chord length of $c = .6$ m is taken into consideration. The entire 2D computational domain is presented in Figure 1 along with various boundary conditions.

4. RESULTS AND DISCUSSION

The external flow around an S809 hydrofoil with .6 m chord length was simulated at $Re = 2.00 \times 10^6$, water density=1000 kg/m$^3$, water viscosity=.0015673 Pa-s and angle of attack 0-14 degree. The obtained pressure coefficients and experimental results for each angle of attack shown in below.

<table>
<thead>
<tr>
<th>Alpha ($\alpha$)</th>
<th>Lift coeff. (C_L)</th>
<th>Drag coeff. (C_D)</th>
<th>Pressure coeff. (C_P)</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.0000</td>
<td>0.088620</td>
<td>0.012022</td>
<td>0.53599</td>
</tr>
<tr>
<td>2.0000</td>
<td>0.32315</td>
<td>0.012143</td>
<td>0.55263</td>
</tr>
<tr>
<td>4.0000</td>
<td>0.55503</td>
<td>0.013694</td>
<td>0.59415</td>
</tr>
<tr>
<td>6.0000</td>
<td>0.77546</td>
<td>0.017194</td>
<td>0.65701</td>
</tr>
<tr>
<td>8.0000</td>
<td>0.97270</td>
<td>0.024207</td>
<td>0.73414</td>
</tr>
<tr>
<td>10.000</td>
<td>1.05793</td>
<td>0.039055</td>
<td>0.78363</td>
</tr>
<tr>
<td>12.000</td>
<td>1.01923</td>
<td>0.058422</td>
<td>0.77931</td>
</tr>
<tr>
<td>14.000</td>
<td>1.04203</td>
<td>0.079898</td>
<td>0.79487</td>
</tr>
</tbody>
</table>
Pressure distribution around a hydrofoil is what generates the lift force exerted on it. High pressure is below the hydrofoil and low pressure is above it. Lift coefficient rises linearly from above Figure 3 as the angle of attack increases. For drag coefficient, increases exponentially with angle of attack. The pressure coefficient first rises as the angle of attack rises, but after some time the pressure coefficient falls and then rises again.
Figure 4 (a to f): $C_p$ vs $x/C$ graphs at different angle of attack

From Figure 4 it can be shown that, the pressure coefficients with different angle of attack along the hydrofoil and validates with Dan M. Somers [5] experimental data. According to both experimental and numerical studies, the pressure difference between the upper and lower surface increases as the angle of attack increases from 0° to 14°. Pressure decreases at the upper surface more quickly and pressure increases at the lower surface more slowly as the angle of attack increases.

Velocity contour, pressure contour and streamlines are plotted at different angle of attack with same Reynolds number in Figure 5. From velocity contours it can be seen that, initially pressure is high and velocity is low. With increasing angle of attack velocity also increases. High angles of attack lead to flow separation. This leads to a sudden decrease in the lift and increased drag because of turbulence and suction region. As it can be seen from pressure contours that, there is a higher pressure area closer to the trailing edge, and since fluids tend to flow from a higher pressure area to a lower pressure area, the fluid flow can even reverse its direction when the angle of attack reaches a certain value. It then recirculates to fill the void and eddy is created. This re-circulation causes a lot of drag. For streamlines along hydrofoils, the shear stresses are acting in the same direction as the flow. They make a large contribution to the drag force, but won’t contribute a significant amount to the lift force.

Figure 5: velocity contours, pressure contours and streamlines along the hydrofoil at angle of attack (0, 14) degrees

5. CONCLUSION

In this study, Finite Volume Method (FVM) has been used to analyze the flow around two-dimensional hydrofoil. The governing equations are expressed in Cartesian velocity components and solution is carried out using SIMPLE algorithm for collocated arrangement. The Spalart-Almaras turbulence model is used to capture solution variables. The results can be summarized as follows:

- The numerical results are found to show excellent agreement with the experimental data until flow begins to separate.
- From pressure contours, it was noticed that there was no flow separation at low angle of attacks but began to separate at high angle of attacks.
• With increasing angle of attack, the values of lift coefficients, drag coefficients of the hydrofoil gradually increased.
• When flow separation occurs, the performance of the Spalart-Almaras turbulence model degrades at high angles of attack.

REFERENCES