

A Comparative Study on Turbulence Models for Simulation of Flow Past NACA 0015 Airfoil Using OpenFOAM

Peerakit Kekina and Chakrit Suwanjumrat

*Department of Mechanical Engineering, Faculty of Engineering, Mahidol University, Salaya, Nakorn Pathom, 73170, Thailand
Laboratory of Computer Mechanics for Design (LCMD), Department of Mechanical Engineering, Faculty of Engineering, Mahidol University, Salaya, Nakorn Pathom, 73170, Thailand*

Abstract. An implementation of C++ language open source code software, OpenFOAM, for simulation of flow past NACA 0015 airfoil was performed to access a suitable turbulence model. Three various turbulence models were selected which comprised of Spalart-Allmaras model, RNG k- ϵ model and Menter SST k- ω model, respectively. The SIMPLE algorithm and Upwind method was used to solve the governing equation to achieve flow solutions of computational fluid dynamic (CFD) models. The flow simulation obtained lift coefficient (C_L) and drag coefficient (C_D) to compare with the wind tunnel experiment data at Reynolds number (Re) of 160,000 and 360,000 with the large range airfoil angle of attack (AOA) from 0 to 20 degree. The suitable CFD model was the Menter SST k- ω model which obtained an average error of C_L and C_D less than 10.96% and 22.21%, respectively.

1 Introduction

Airfoil simulation referred to predict aerodynamics performance which was the active force acting on airfoil. Computational Fluid Dynamics (CFD) were the methodology to study phenomenon of fluid flow past airfoil with the benefit to reduce time and cost of physical experiments. The most of airfoil simulation was analyzed under the turbulent region. The precise simulation solution should be obtained when grid generation fined around wall region. The non-dimension y^+ of first cell for CFD techniques was controlled into viscous sub-layer less than five [1] but some researcher had suggested one [2].

Reynolds-averaged Navier-Stokes equation (RANS) was always employed into CFD techniques which focused on the mean flow of turbulence. An extra term, Reynolds stresses, was estimated by turbulence models classified by number of additional term into the transport equation. The one transport equation, Spalart-Allmaras (S-A) model was developed to calculate kinematics eddy viscosity parameter and length scale in the term of local mean vorticity [3]-[5]. Two additional terms were used to estimate Reynolds stresses such as standard k- ϵ model, RNG k- ϵ model [6], Wilcox k- ω model and Menter shear stress transport (SST) k- ω model [7]. In an external aerodynamics problem, Spalart-Allmaras model, Wilcox k- ω model and Menter SST k- ω model were suggested for simulation. There are many researchers used commercial CFD software such as ANSYS, FLUENT and STAR-CD which contained turbulence models [8] [9]. Unfortunately they were limited on an expensive

license cost. The Open Source Field Operation and Manipulation (OpenFOAM) software had been used C++ language for CFD code without license cost under GNU General Public License [10]. This research would apply turbulence models using OpenFOAM to determine an appropriate model for airfoil simulations and wind turbine blade design which was an external aerodynamics problem in a further work.

2 Turbulence models

In this research, three turbulence models comprised of Spalart-Allmaras (S-A) model, Menter SST k- ω model and RNG k- ϵ model were implemented to simulate flow past an airfoil.

2.1 Spalart-Allmaras model

The S-A model is form with the transport equation of the kinematic eddy viscosity ($\tilde{\nu}$). The one-equation of the S-A model is written by:

$$\frac{\partial(\rho\tilde{\nu})}{\partial t} + \text{div}(\rho\tilde{\nu}U_i) = \frac{1}{\sigma_\nu} \text{div} \left[(\mu + \rho\tilde{\nu}) \nabla\tilde{\nu} + C_{b2}\rho \frac{\partial\tilde{\nu}}{\partial x_k} \frac{\partial\tilde{\nu}}{\partial x_k} \right] + C_{b1}\rho\tilde{\nu}\tilde{\Omega} - C_{w1}\rho \left(\frac{\tilde{\nu}}{\kappa y} \right)^2 f_w \quad (1)$$

where $\tilde{\nu}$ is the kinematic eddy viscosity, f_w is the wall damping function, μ is the dynamics viscosity. The constant include σ_ν , C_{b1} , C_{b2} and κ has value of 0.67, 0.1355, 0.622 and 0.4187, respectively.

2.2 Menter SST k- ω model

The Menter SST k- ω has been developed from the Wilcox k- ω [11] to precise simulation results on the boundary layer. Two equations of the model can be written by:

$$\frac{\partial(\rho k)}{\partial t} + \text{div}(\rho k U_i) = \text{div} \left[\left(\mu + \frac{\mu_t}{\sigma_k} \right) \nabla k \right] + P_k - \beta^* \rho k \omega \frac{k}{\omega} \quad (2)$$

$$\begin{aligned} \frac{\partial(\rho \omega)}{\partial t} + \text{div}(\rho \omega U_i) = & \text{div} \left[\left(\mu + \frac{\mu_t}{\sigma_{\omega,1}} \right) \nabla \omega \right] \\ & + \gamma_2 \left(2\rho S_{ij} S_{ij} - \frac{2}{3} \rho \omega \frac{\partial U_i}{\partial x_j} \delta_{ij} \right) \\ & - \beta_2 \rho \omega^2 + \frac{2\rho}{\sigma_{\omega,2} \omega} \frac{\partial k}{\partial x_k} \frac{\partial \omega}{\partial x_k} \quad (3) \end{aligned}$$

where P_k is the rate of production of turbulent kinetic energy. The constant includes σ_k , β^* , $\sigma_{\omega,1}$, γ_2 , β_2 and $\sigma_{\omega,2}$ has value of 1.00, 0.09, 2.0, 0.44, 0.083 and 1.17 respectively.

2.3 RNG k- ϵ model

$$\frac{\partial(\rho k)}{\partial t} + \text{div}(\rho k U_i) = \text{div} \left[\alpha_k \mu_{\text{eff}} \nabla k \right] + \tau_{ij} \cdot S_{ij} - \rho \epsilon \quad (4)$$

$$\begin{aligned} \frac{\partial(\rho \epsilon)}{\partial t} + \text{div}(\rho \epsilon U_i) = & \text{div} \left[\alpha_\epsilon \mu_{\text{eff}} \nabla \epsilon \right] + C_1 \frac{\epsilon}{k} \tau_{ij} \cdot S_{ij} \\ & - C_2 \rho \frac{\epsilon^2}{k} \quad (5) \end{aligned}$$

where k is the turbulent kinetic energy, ϵ is the turbulent dissipation. The constant include C_μ , α_ϵ , C_1 and C_2 has value of 0.0845, 1.39, 1.42 and 1.68 respectively.



Figure 1. The NACA 0015 airfoil installing in the wind tunnel.

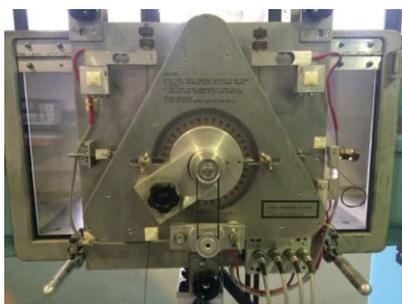


Figure 2. Triangular load cell for measuring lift force and drag force.

3 Wind tunnel experiment

The airfoil profile, NACA0015, with chorded length of 190 mm and span length of 285 mm was tested in the wind tunnel model WT300 (Fig. 1). Lift force and drag force were measured using the triangular load cell as shown in Fig. 2. The airfoil angle has been adjusted from 0 to 20 degree by using a load cell spindle. The flow velocity was controlled by an axial fan to generate Reynolds number (Re) of flow past airfoil at 160,000 and 360,000.

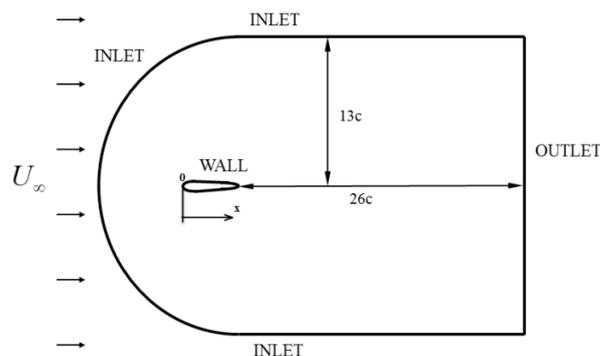


Figure 3. The C-type domain for airfoil simulation.

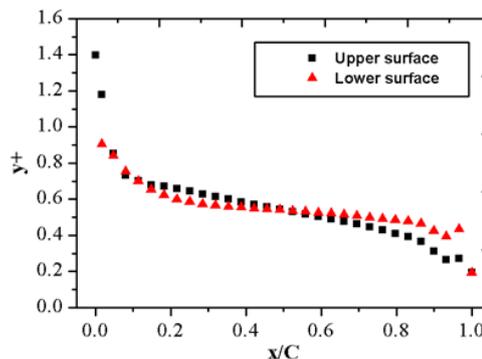


Figure 4. The y^+ distribution on airfoil surface.

4 Computational fluid dynamics

The computational domain has selected by using the C-type domain which radius and downstream length are 13 and 26 times of a chord length respectively (Fig. 3). The nearest cells or grids on airfoil were attempted to control by y^+ value to be less than 1 which less than the satisfied values as equal to 11.63 [1]. The y^+ value around airfoil is plotted against distance from leading edge to tailing edge as shown in Fig. 4. The cell generation is performed simultaneously after y^+ controlling, then the number of nodes around the airfoil is variable from 100 to 600 which made cells being into a simulated domain from 19,818 to 100,621 respectively. The steady result of lift coefficient (C_L) and drag coefficient (C_D) of airfoil at the angle of attack (AOA) of 4 degree is the cell independent test by the node number of 300 around airfoil and the total cell of 58,454. Fig. 5 shows a cell structure of an

airfoil simulation at the near wall region with y^+ between 0.2 and 1.4. The SIMPLE algorithm was used to solve the governing equation of airfoil simulation, while pressure-velocity coupling problem used Upwind method.

Boundary condition was inlet flow from the left side of the CFD domain which assigned uniform velocities and outlet flow was set by an atmospheric pressure. Front and back domain boundary were set by empty type. No slip condition and no wall function were set on wall boundary. An incompressible flow was used for air with density (ρ) and a dynamics viscosity (μ) of 1.225 kg/m^3 and $1.8375 \times 10^{-5} \text{ kg/(m s)}$ respectively.

5 Results and discussion

The C_L and C_D achieved by calculation with simulation results of three turbulence models. The simulation and experiment result of flow past airfoil with Reynolds number (Re) of 160,000 was compared by graphs in Fig. 6 and Fig. 7. The lift force increased linearly until the AOA was 10 degree and decreased at 12 degree. On the other hand drag force was opposite to the lift force along the AOA. The RNG $k-\epsilon$ model had trend of the C_L graph in agreement with experimental data more than the other turbulence models. Subsequently, the SST $k-\omega$ model had C_D graph in a good agreement with experimental data more than the other turbulence models. The experimental data had shown the stall angle at 10 degree of AOA while the S-A and RNG $k-\epsilon$ model had shown at 14 and 12 degree respectively.

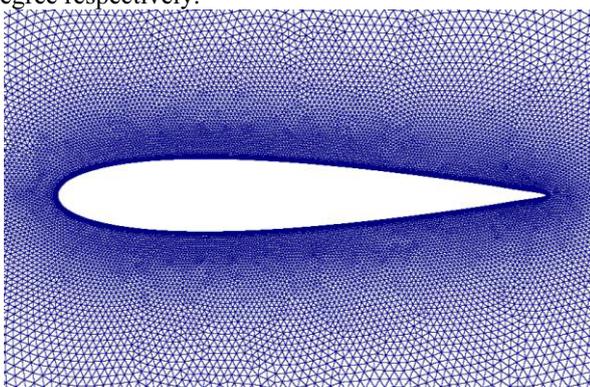


Figure 5. Cells arrangement on near wall region of airfoil by y^+ controlling.

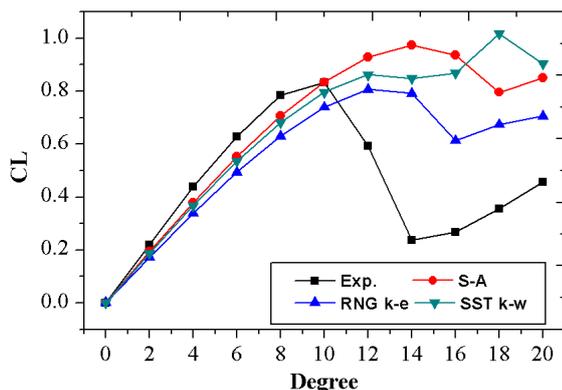


Figure 6. Lift coefficient on airfoil with various AOA by Re of 160,000.

The simulation with Re of 360,000 had predicted C_L and C_D and had been achieved the same behaviour results by using Re of 160,000. Fig. 8 and 9 show the comparison C_L and C_D graph between CFD and the physical experiment under Re of 360,000 respectively. The SST $k-\omega$ model had trend of the C_L and C_D graph in a good agreement with experimental data than the other turbulence models. The stall angle of the S-A, SST $k-\omega$ and RNG $k-\epsilon$ model had shown at 12 degree.

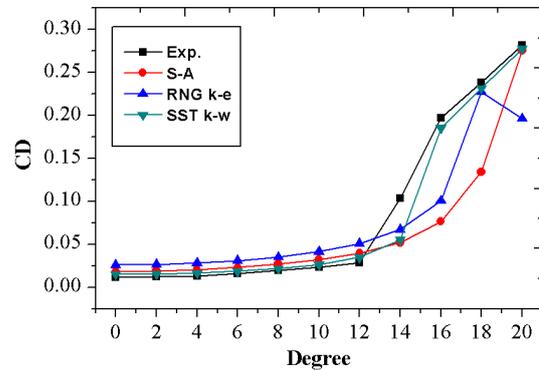


Figure 7. Drag coefficient on airfoil with various AOA by Re of 160,000.

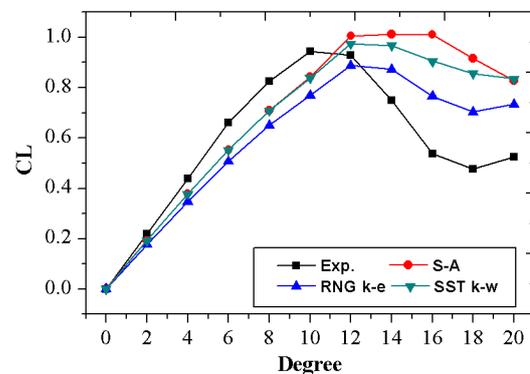


Figure 8. Lift coefficient on airfoil with various AOA by Re of 360,000.

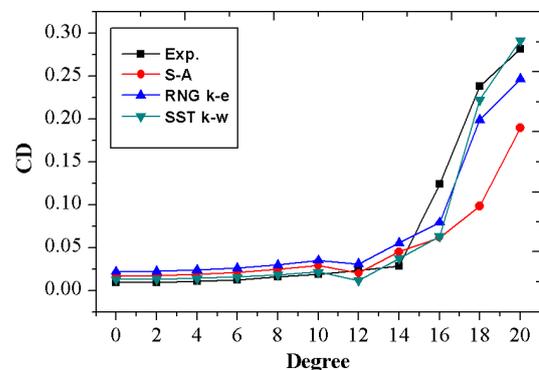


Figure 9. Drag coefficient on airfoil with various AOA by Re of 360,000.

Table 1. The average error of turbulence models in the AOA range of 0-10 degree.

Model	AOA 0-10 degree (%)			
	Re = 160,000		Re = 360,000	
	C_L	C_D	C_L	C_D
S-A	8.01	48.92	11.18	71.15
Menter SST $k-\omega$	10.61	21.25	11.30	28.28
RNG $k-\epsilon$	16.13	102.50	17.24	114.78

The result of CFD simulation had average error when compared with experimental data that described in Table I and Table II for the AOA range of 0-10 and 11-20 degree respectively. The S-A and Menter SST $k-\omega$ model were both in good agreement with experimental data but the Menter SST $k-\omega$ model had results close to experiment data. The velocity of flow and stream line past airfoil at AOA of 12 degree under the Re of 360,000 by using the Menter SST $k-\omega$ model shows in Fig. 10. The color contour expressed the flow velocity. The maximum velocity was red and the minimum velocity was blue. The maximum velocity was about 8 m/s happened on the leading edge of airfoil when it was set AOA less than 12 degree. The solid line showed the stream line of the air flow. The wake had occurred on the tailing edge of airfoil distinctly after AOA was more than 10 degree. The stream line of flow past airfoil happened vortex on the trailing edge at AOA of 12 degree.

Table 2. The average error of turbulence models in the AOA range of 11-20 degree.

Model	AOA 11-20 degree (%)			
	Re = 160,000		Re = 360,000	
	C_L	C_D	C_L	C_D
S-A	165.61	39.64	56.06	43.02
Menter SST $k-\omega$	162.35	16.24	48.07	28.17
RNG $k-\epsilon$	108.81	39.91	30.07	38.57

The pressure contours around airfoil by simulation at the same AOA in Fig. 10 are shown in Fig. 11. The color contour expressed the pressure around the airfoil. The maximum pressure was 80 Pa happened on lower surface contour expressed the pressure around the airfoil. The maximum pressure was 80 Pa happened on lower surface of the airfoil which depicted by red. The maximum pressure as the drag force happened at AOA of 12 degree (Fig. 11).

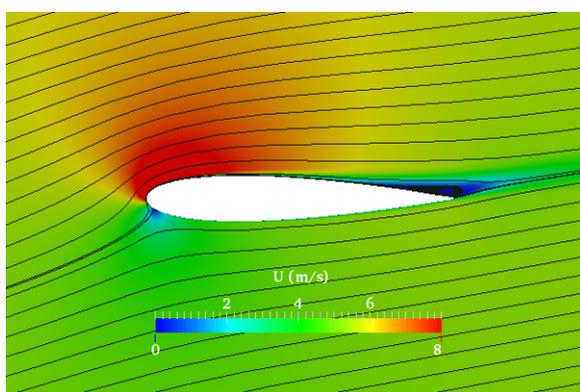


Figure 10. Streamline past airfoil at AOA of 12 degree by using the Menter SST $k-\omega$ model.

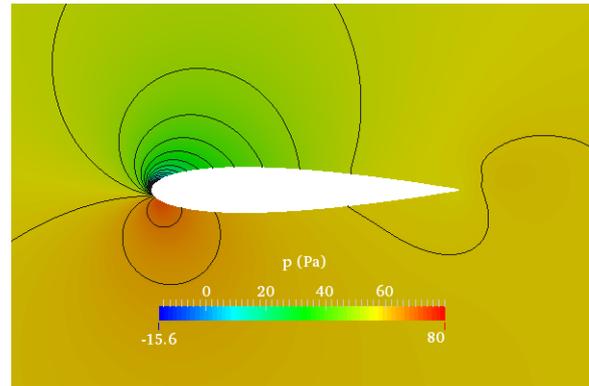


Figure 11. Pressure contour around airfoil at AOA of 12 degree by using the Menter SST $k-\omega$ model.

6 Conclusions

An implementation of turbulence models for airfoil simulation using open source software had been performed. The CFD simulation obtained results to compare with wind tunnel experiment. The lift and drag coefficient were employed to determine the suitable turbulence model. The Menter SST $k-\omega$ was a suitable model for the airfoil simulation at AOA before stall region (0-10 degree). The stream line formed to be vortex after stall region (12 degree) distinctly. For the meanwhile, the RNG $k-\epsilon$ model was suitable for stall region (11-20 degree). Nevertheless, the Menter SST $k-\omega$ model was recommended for studying and designing airfoil of wind turbines because the least average error was obtained in both regions.

References

1. H. K. Versteeg, W. Malalasekera, *An Introduction to Computational Fluid Dynamics: The Finite Volume Method*, second ed., Prentice Hall, England, (2007).
2. F. Villalpando, M. Reggio, A. Ilinca, *Model. Simul. Eng.* **2011** (2011)
3. P.R. Spalart, S.R. Allmaras, *La Recherche aérospatiale.* **1** (1994)
4. P.R. Spalart, S.R. Allmaras, *A one-equation turbulence model for aerodynamic flow*, AIAA, Washington D.C., (1992)
5. V. Yakhot, S.A. Orszag, S. Thangam, T.B. Gatski, C.G. Speziale, *Phys. Fluids A.* **4**, 7, (1992)
6. K. nordanger, R. Haldahl, T. Kvamsdal, A.M. Kvaving, A. Rasheed, *Comput. Methods Appl. Mech. Engrg.* **290** (2015)
7. F.R. Menter, *AIAA J.* **32**, 8, (1994)
8. R. Belamadi, A. Djemili, A. Ilica, R. Mdouki, *J. Wind Eng. Ind. Aerod.* **112**, 146 (2016)
9. M.H. Gardiri, N.L.N. Ibrahim, M.F. Mohamed, *Engineering Journal* **17**, 9, (2013)
10. OpenFOAM, *The Open Source CFD Toolbox User Guide Version 2.3.0* (2014)
11. D.C. Wilcox, *Turbulence modeling for CFD*, DCW Industries, Inc. (1993)