Penerbit Akademia Baru

CFD Letters

Journal homepage: www.akademiabaru.com/cfdl.html
ISSN: 2180-1363

Comparing the Effect of Different Turbulence Models on The CFD Predictions of NACA0018 Airfoil Aerodynamics

Open Access

Sher Afghan Khan[1,*], Musavir Bashir[2], Maughal Ahmed Ali Baig[3], Fharukh Ahmed Ghasi Mehaboob Ali[4]

[1] Department of Mechanical Engineering, Faculty of Engineering, IIUM, Malaysia
[2] School of Aerospace Engineering, USM, Penang, Malaysia
[3] Department of Mechanical Engineering, CMR Technical Campus, Hyderabad, India
[4] Department of Mechanical Engineering, Government Engineering College, Huvinahadagali, Bellary dist., Karnataka, India

ARTICLE INFO

Article history:
Received 22 January 2020
Received in revised form 17 March 2020
Accepted 22 March 2020
Available online 28 March 2020

ABSTRACT

The analysis of aerodynamic forces on two-dimensional airfoils and flow phenomena associated with different Reynolds number has been widely investigated for aerospace vehicles and wind turbines. Various numerical methods have been used with different turbulence models, and the discrepancy in flow physics differs between each model. This work presents the numerical analysis of the aerodynamic performance of NACA 0018 airfoil using different turbulence models. The Computational Fluid Dynamics (CFD) solver, Ansys Fluent, is used for this numerical study. The available experimental data of NACA 0018 airfoil is used for comparing numerical outputs, and the difference in the lift, drag, and pressure coefficients are evaluated, respectively. Structured meshing is employed for all the investigated models for the analysis for lift and drag coefficients using four different turbulence models. Measurements of aerodynamic coefficients and surface pressures are recorded for a range of Re=0.8x10^5 to Re=0.3x10^6 and angles of attack (0° to 18°). The SST k-omega model provided the best lift coefficient predictions for low angles of attack Results also indicate that the flow separation regions and reattachment locations can be predicted by the Transition k-kl-ω model.

Keywords:
Lift Coefficient; Pressure Coefficient; Low Reynolds Number; Turbulence Models; Flow Separation

1. Introduction

Airfoils are essential to numerous engineering applications and can be found in sizes ranging from the wings of Unmanned Aerial Vehicles and the blades of wind turbines to the wings of large commercial aircraft [1,2]. Further, airfoils operate over a range of speeds, from slow gliders to supersonic military aircraft. These broad areas of applications over which airfoils are employed presents a significant challenge to engineers in terms of required airfoil performance. Thus numerous

* *Corresponding author.*
E-mail address: sakhan@iium.edu.my (Sher Afghan Khan)

https://doi.org/10.37934/cfdl.12.3.110

airfoil designs have been analyzed at a wide range of operating conditions is the Reynolds number for different aerodynamic performances [3-5]. One of the challenging design and aerodynamic analysis of airfoils is at low Reynolds numbers.

It is primarily recognized that there is no universal model that eventually pronounces the complete characteristics of fluid flow and its interactions with airfoils with reasonable accuracy while employing a reasonable amount of computational resources. This modeling problem becomes more complex as more physical phenomena are considered (e.g., if turbulent, compressible, and multiphase flows are considered, among other relevant conditions). Many experimental investigations and numerical simulations have been conducted to assess the aerodynamic coefficients and the boundary layer characteristics [6-8]. Earlier researchers have shown that both angles of incidence and the Reynolds number have got a substantial effect on the aerodynamic performance of 2-D aerofoils [9-11]. McCroskey [12] wrote a review paper for wind tunnel tests conducted and validated with the CFD, and the results compared showed good accuracy.

Researchers have conducted the experiments on NACA 0018 airfoil at low Reynolds number [13-15]. One such investigation has estimated the aerodynamic coefficients for up to 30^0 angle of attack and a wide range of Reynolds number [13]. Another investigation recorded the pressure coefficient and the lift coefficient over the aerofoil from 0^0 to 16^0 degrees of angle of attack for different Reynolds numbers [14]. For NACA 2412 airfoil using C-type of the fluid domain in order to analyses the fluid flows with incompressible flow has been studied and shows that the lift and drag coefficient variation using the different angle of attack [16]. Furthermore, the CFD analysis on different objects such as non-circular cylinder [17,18], splitter plate [19], wedge [20,21], and CD nozzle [22-27] has been studied well using the suitable turbulence model through ANSYS fluent.

The literature has shown that there is a significant discrepancy in the numerical results obtained by different turbulence models [25]. There is a need to understand the effect of different turbulence models on the aerodynamic analysis of an airfoil. The different flow modeling methodologies and the accuracy of results should be studied. The aim of the present work is to carry out the numerical analysis of the aerodynamic performance of NACA 0018 airfoil using different turbulence models. The Computational Fluid Dynamics (CFD) solver, Ansys Fluent, is used for this numerical study.

2. Methodology

In this paper, the NACA 0018, the well-documented airfoil from the 4-digit series of NACA airfoils, was used. The NACA 0018 airfoil is symmetrical; the 00 indicates that it has no camber. The 18 indicates that the airfoil has an 18% thickness to chord length ratio; it is 18% as thick as it is long. Reynolds number for the simulations range from $Re=0.8x10^5$ to $Re=0.3x10^6$, same with the reliable experimental data from NACA (1959), in order to validate the present simulation. The free stream temperature is 300 K, which is the same as the environmental temperature. The density of the air at the given temperature is ρ = 1.225kg/m3, and the viscosity is μ = 1.7894×10-5kg/ms. The SST k-omega turbulence model was used in ANSYS Fluent 16. Calculations were done for angles of attack ranging from 0 to 18°. The airfoil profile, boundary conditions, and meshes were all created in the Workbench. The inlet, outlet and wall boundary conditions are depicted in Figure 1. The resolution of the mesh was more magnificent, close to the airfoil boundary layer regions where greater computational accuracy was needed.

The majority of time spent on a CFD project in the industry is usually devoted to successfully generating a mesh for the domain geometry that allows a compromise between desired accuracy and solution cost. After the creation of the grid, a solver is able to solve the governing equations of the problem. The essential procedural steps for the solution of the problem are the following. First, the modeling goals have to be defined, and the model geometry and grid are created. Then, the solver and the physical models are stepped up in order to compute and monitor the solution. Afterward, the results are examined and saved, and if it is necessary, we consider revisions to the numerical or physical model parameters.

3. Governing Equation

For the two-dimensional, unsteady and incompressible flow, we consider that the governing equations are the RANS equations as follows;

$$\frac{\partial u_i}{\partial x_i} = 0 \tag{1}$$

$$\frac{\partial u_i}{\partial t} + u_j \frac{\partial u_i}{\partial x_j} = -\frac{1}{\rho}\frac{\partial p}{\partial x_i} + v\frac{\partial^2 u_i}{\partial x_j \partial x_j} - \frac{\partial u_i' u_j'}{\partial x_j} \tag{2}$$

where ui is the mean velocity, ν the kinematic viscosity of the air, q the density of air, ρ the pressure, and $-u\vec{i}u\vec{j}$ the Reynolds stress.

4. Turbulence Models
4.1 The SST k-omega Turbulence Model

The k-omega SST turbulence model includes transport of the turbulence shear stress in the definition of the turbulent viscosity. These features make this model more accurate and reliable for a more comprehensive class of flows (for example, adverse pressure gradient flows, aerofoils, and transonic shock waves) than the standard and the BSL - models. So, that is the main reason behind choosing the k-omega SST turbulence model over standard and BSL model. K-omega SST turbulence model is governed by,

$$\frac{\partial(\rho k)}{\partial t} + \frac{\partial(\rho U_j k)}{\partial x_j} = P_k - D_k + \frac{\partial}{\partial x_j}\left[(\mu + \sigma_k \mu_t)\frac{\partial k}{\partial x_j}\right] \tag{3}$$

$$\frac{\partial(\rho \omega)}{\partial t} + \frac{\partial(\rho U_j \omega)}{\partial x_j} = P_\omega - D_\omega + \frac{\partial}{\partial x_j}\left[(\mu + \sigma_\omega \mu_t)\frac{\partial \omega}{\partial x_j}\right] + 2(1 - F_1)\frac{\rho \sigma_{\omega 2}}{\omega}\frac{\partial k}{\partial x_j}\frac{\partial \omega}{\partial x_j} \tag{4}$$

Where β*=β^*=$\varepsilon/k\omega$ and the turbulence stress tensor is

$$\tau_{ij} = -\rho u_i' u_j' = \mu_t\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} - \frac{2}{3}\frac{\partial u_k}{\partial x_k}\delta_{ij}\right) - \frac{2}{3}\rho k \delta_{ij} \tag{5}$$

The turbulence viscosity can be estimated byvt=a_1k/max ($a_1\omega$,ΩF_2), where Ω is the absolute value of the vorticity, a_1 = 0.31.

*4.2 The k-kl-**ω** Model*

Transition flows work on K-kl-ω model as the accuracy levels predicted by this model are high. Three expressions are modeled for finding viscosity in turbulent mode.

$$\frac{Dk_T}{D} = P_{KT} + R + R_{NAT} - \omega_{kT} + \frac{\partial}{\partial x_j}\left[\left(v + \frac{a_T}{a_k}\right)\frac{\partial k_T}{\partial x_j}\right] \quad (6)$$

$$\frac{Dk_L}{D_t} = P_{KL} - R - R_{NAT} - D_L + \frac{\partial}{\partial x_j}\left[v\frac{\partial k_L}{\partial x_j}\right] \quad (7)$$

$$\frac{D_\omega}{D_t} = C_{\omega 1}\frac{\omega}{k_T}P_{KT} + \left(\frac{C_{\omega R}}{f_W} - 1\right)\frac{\omega}{k_T}(R + R_{NAT}) - C\omega_2\omega^2 + C\omega_3 f_\omega a_T f_W^2\frac{\sqrt{k_T}}{d^3} + \frac{\partial}{\partial x_j}\left[\left(v + \frac{a_Y}{a_\omega}\right)\frac{\partial \omega}{\partial x_j}\right] \quad (8)$$

5. Numerical Analysis

The data formulated by Gerakopulos *et al.,* [6] is used for testing turbulence models on NACA 0018 airfoil. ANSYS Design Modular is used for the generation of the surface around the airfoil, and the same is depicted in Figure 2.

5.1 Computational Domain and Meshing

The solution domain for the NACA0018 airfoil is generated by using ANSYS/Workbench. The solution domain is the C-shaped domain, as shown in Figure 1, and the resulted mesh consists of structured quad elements, as shown in Figure 2. The initial values of viscosity, pressure, and density are taken as that of sea level.

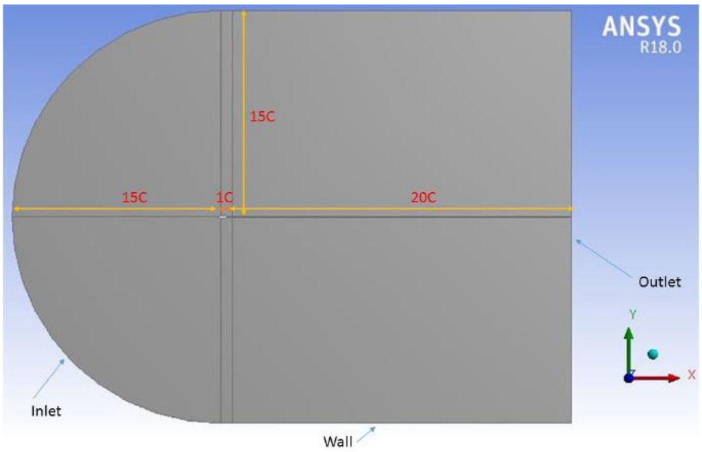


Fig. 1. Domain of 2-D airfoil geometry

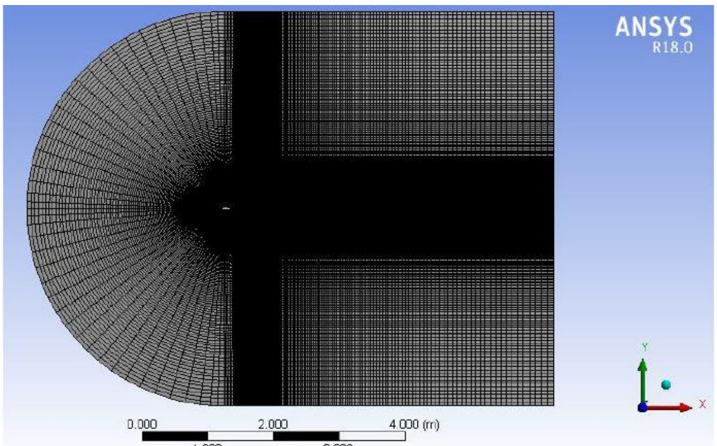


Fig. 2. Meshing of the 2D airfoil

5.2 Grid Independence Test

They were utilizing mid-fine as well as a superfine mesh, which is more suited for the optimized values of the drag and lift. The residuals were monitored as a measure of solution convergence. Residuals directly quantify the error in the solution, and it measures the local imbalance of a conservative variable in each control volume. In this study, the residual value setting value considered was 1e-6. Base on the grid independence tests, it is concluded that mid-fine mesh seems to be optimum for the present study in order to obtain the optimum value of the lift and drag coefficients. Hence, Table 1 indicates the results of various mesh configurations and under what conditions they become independent.

Table 1
Mesh Independence test results

Mesh	CL	CD
Standard	0.5827	0.0373
Coarse	0.6221	0.0313
Mid	0.7021	0.0207
Mid-Fine	0.7172	0.0213
Fine	0.7172	0.0213

6. Results

The presented results include Reynolds number of $0.8 x 10^5$ to $0.3 x 10^6$ and the angle of attack up to 18^0. In this study, the results obtained from the numerical simulations using different turbulence models were compared with experimental results from the literature. Under the standard situations, the evaluation of lift and drag coefficients are presented in Table 2, and the standard SST k-ω model has given good results after the results were compared with the experimental results from the literature. Therefore, this model is selected for the simulations at all angles of attack and different Reynolds number.

Table 2
Computational results at different turbulence models

Turbulence Model	C_L	C_D
Standard k-ε	0.8171	0.0234
Standard k-ω	0.94129	0.0201
SST-k-ω	1.019	0.0193
Transition k-kl-ω	1.05677	0.0187

However, the SST k-ω was not able to predict the flow separation phenomena. Therefore, the study was carried using the Transition k-kl-ω model. This turbulence model chosen was able to predict the separation bubble and flow behavior at high angles of attack. It should be noted that the other models, such as Standard k- ε or Standard and k-ω model was also able to predict the lift coefficients at low angles of attack, as shown in Figure 3.

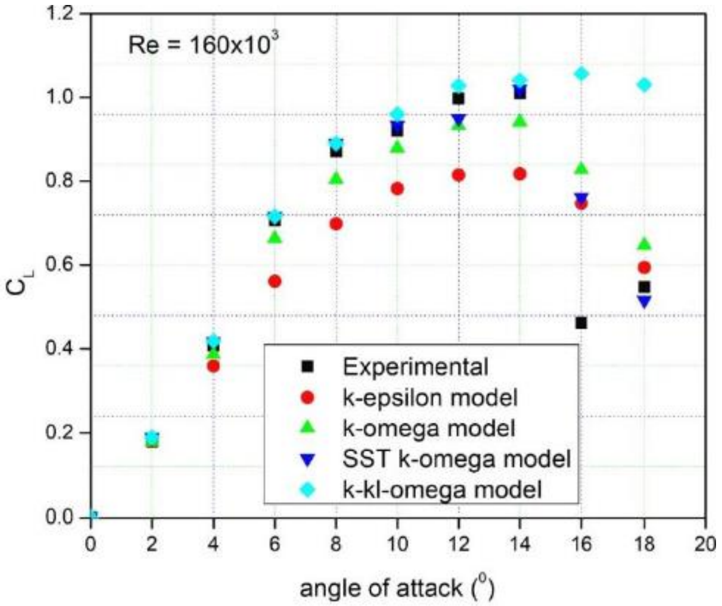
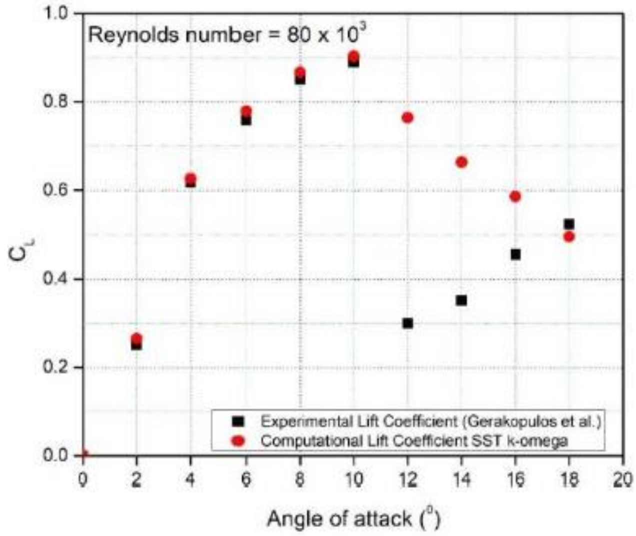


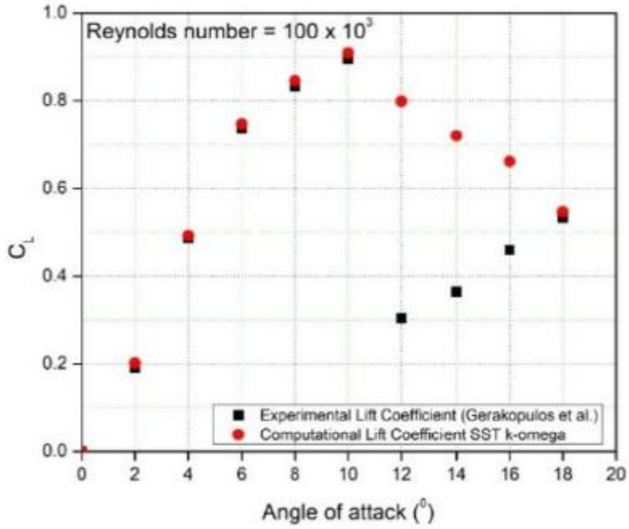
Fig. 3. Lift coefficient for different turbulence models at Re = 160×10^3

6.1 Lift Coefficient Results

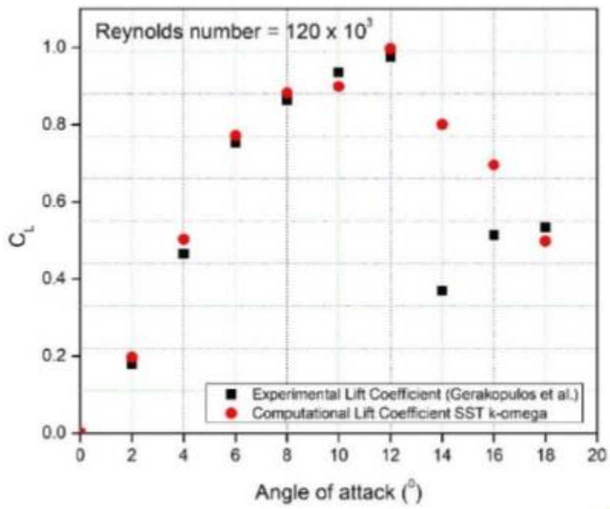
Lift coefficients obtained from the computational studies at different angles of attack and Reynolds number are exhibited in Figure 4(a) to 4(f). The outcomes demonstrate that, usually, the stall angle increases with increasing the Reynolds number. An increase in the Reynolds number from Re = 80×10^3 to Re = 200×10^3 results in an increase in the stall angle from AOA of 10° till 14° and an increase in stall for higher lift coefficient.



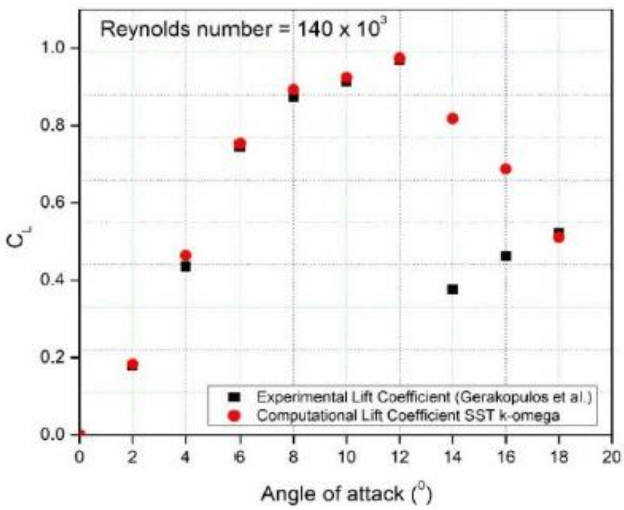
4(a)



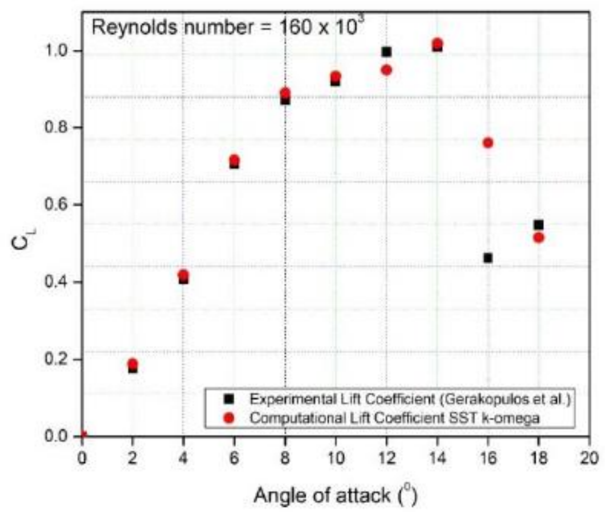
4(b)



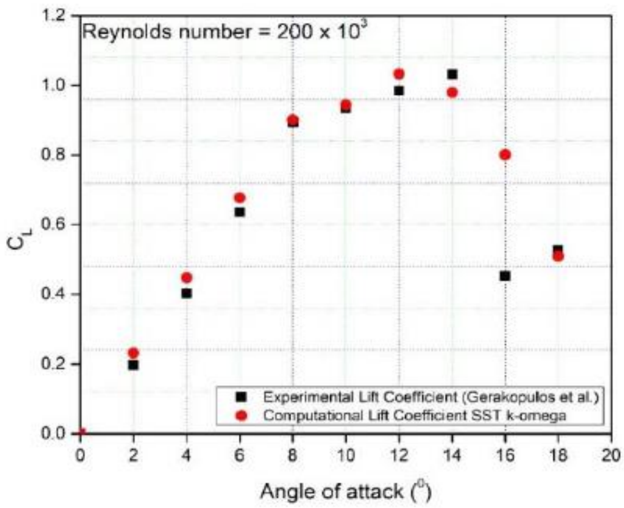
4(c)



4(d)



4(e)

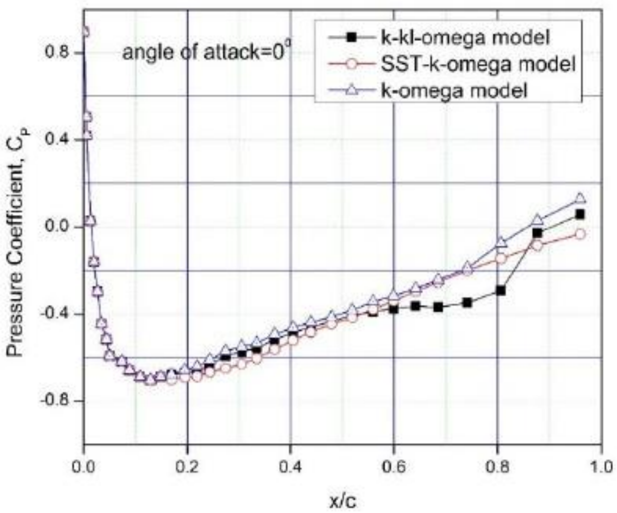


4(f)

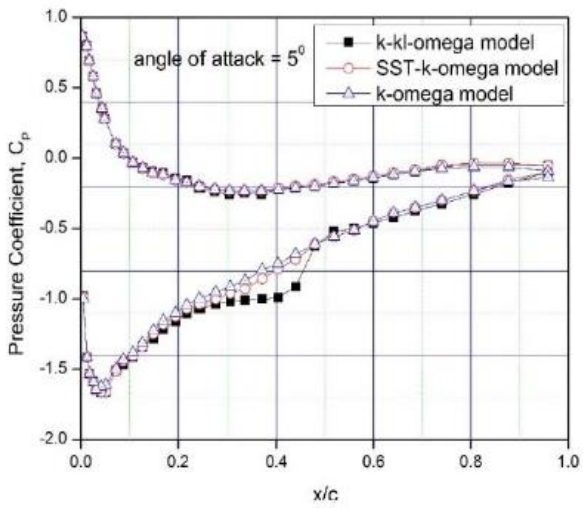
Fig. 4. Lift coefficient for different Turbulence Models

6.2 Pressure Coefficient Distribution Results

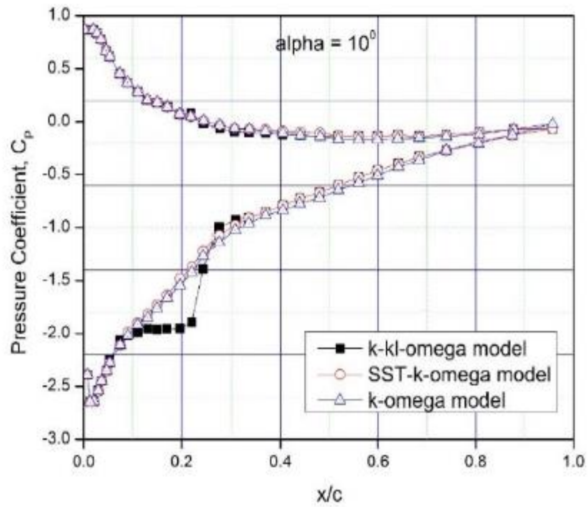
From the results, it is clear that the flow separation can be predicted only by the Turbulent Transition model. Figures 5(a) to 5(d) shows that the laminar separation bubble moves over the airfoil surface at different angles of attack. The result of k-ω and SST k-ω models are also plotted in the same plot, and they were not able to predict the flow separation behavior. However, k-kl-***ω*** Model provides excellent results as it predicts the flow behavior both at different angles of attack accurately, as shown in Figures 5(a) to 5(d).



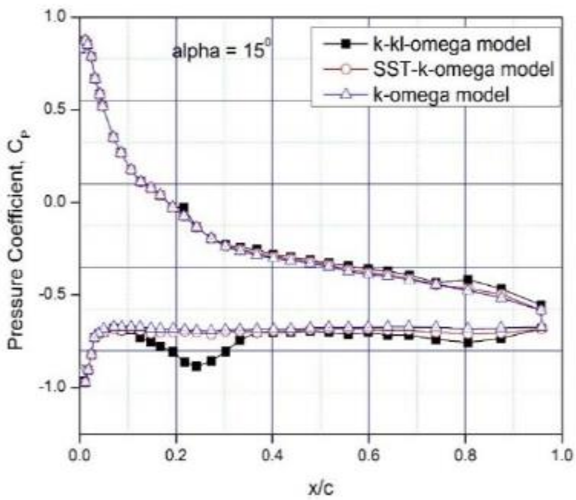
5(a)



5(b)



5(c)



5(d)

Fig. 5. Pressure Distribution of different turbulent models compared

Therefore, the overall aerodynamic results obtained in this study reveal that reliable lift coefficients are obtained using standard SST k-ω model, and the resulted values of the experimental study and the current CFD study are found to be in good agreement. However, an accurate and transition turbulence model is needed to capture capturing the transition behavior for low Reynolds numbers flows. In this study, k-kl-***ω*** Model was able to predict the flow separation.

7. Conclusions

The results obtained from the numerical simulations using the FLUENT has demonstrated its capability to obtain reliable results similar to the wind tunnel results. The Ansys Fluent provides reliable results, and the researchers can save the enormous cost involved in the wind tunnel test.

The lift and drag results obtained from the standard SST k-ω turbulence model have given accurate results even at low Reynolds numbers. The SST k-ω turbulence model is best for predicting lift coefficients, but drag coefficients show discrepancy when compared with experimental results. Similarly, the results of surface pressure coefficients are plotted and compared with experimental results. But the SST k-ω turbulence model fails to predict the separation bubbles. The transition can be predicted only at high angles of attack. To predict the flow separation phenomena, k-kl-ω Model was capable of predicting the separation bubble, overall correct lift, and drag coefficient at a low angle of attack.

References

[1] Samion, Siti Ruhliah Lizarose, and Mohamed Sukri Mat Ali. "Aerodynamic noise measurement in anechoic wind tunnel of rod-airfoil with leading edge serrations." *Journal of Advanced Research in Fluid Mechanics and Thermal Sciences* 47, no. 1 (2018): 97-107.

[2] Rojratsirikul, Pinunta, Zhijin Wang, and Ismet Gursul. "Unsteady fluid–structure interactions of membrane airfoils at low Reynolds numbers." *Experiments In Fluids* 46, no. 5 (2009): 859. https://doi.org/10.1007/s00348-009-0623-8

[3] Van Treuren, Kenneth W. "Small-scale wind turbine testing in wind tunnels under low Reynolds number conditions." *Journal of Energy Resources Technology* 137, no. 5 (2015). https://doi.org/10.1115/1.4030617

[4] Winslow, Justin, Hikaru Otsuka, Bharath Govindarajan, and Inderjit Chopra. "Basic Understanding of Airfoil Characteristics at Low Reynolds Numbers (10 4–10 5)." *Journal of Aircraft* 55, no. 3 (2018): 1050-1061. https://doi.org/10.2514/1.C034415

[5] Shen, Xiang, Eldad Avital, Gordon Paul, Mohammad Amin Rezaienia, Pihua Wen, and Theodosios Korakianitis. "Experimental study of surface curvature effects on aerodynamic performance of a low Reynolds number airfoil for use in small wind turbines." *Journal of Renewable and Sustainable Energy* 8, no. 5 (2016): 053303. https://doi.org/10.1063/1.4963236

[6] Kamada, Yasunari, Takao Maeda, Junsuke Murata, and Yusuke Nishida. "Effect of turbulent inflows on airfoil performance for a Horizontal Axis Wind Turbine at low Reynolds numbers (Part II: Dynamic pressure measurement)." *Energy* 112 (2016): 574-587. https://doi.org/10.1016/j.energy.2016.06.126

[7] Gerakopulos, Ryan, Michael Boutilier, and Serhiy Yarusevych. "Aerodynamic characterization of a NACA 0018 airfoil at low Reynolds numbers." In *40th Fluid dynamics conference and Exhibit*, p. 4629. 2010. https://doi.org/10.2514/6.2010-4629

[8] Hansman Jr, R. John, and Anthony P. Craig. "Low Reynolds number tests of NACA 64-210, NACA 0012, and Wortmann FX67-K170 airfoils in rain." *Journal of Aircraft* 24, no. 8 (1987): 559-566. https://doi.org/10.2514/3.45476

[9] Aono, Hikaru, Masayuki Anyoji, Daiki Hamada, Shotaro Wakui, and Tomoaki Tatsukawa. "A study on development of airfoil shape toward low Reynolds-number dependence of aerodynamic characteristics under low-Reynolds-number-flow conditions." In *2018 AIAA Aerospace Sciences Meeting*, p. 1085. 2018. https://doi.org/10.2514/6.2018-1085

[10] Lian, Yongsheng, and Wei Shyy. "Laminar-turbulent transition of a low Reynolds number rigid or flexible airfoil." *AIAA Journal* 45, no. 7 (2007): 1501-1513. https://doi.org/10.2514/1.25812

[11] O'Meara, M. M., and Thomas J. Mueller. "Laminar separation bubble characteristics on an airfoil at low Reynolds numbers." *AIAA Journal* 25, no. 8 (1987): 1033-1041. https://doi.org/10.2514/3.9739

[12] McCroskey, W. J. *A critical assessment of wind tunnel results for the NACA 0012 airfoil.* No. NASA-A-87321. NATIONAL AERONAUTICS AND SPACE ADMINISTRATION MOFFETT FIELD CA AMES RESEARCHCENTER, 1987.

[13] Timmer, W. A. "Two-dimensional low-Reynolds number wind tunnel results for airfoil NACA 0018." *Wind Engineering* 32, no. 6 (2008): 525-537. https://doi.org/10.1260/030952408787548848

[14] Nakano, T., N. Fujisawa, Y. Oguma, Y. Takagi, and S. Lee. "Experimental study on flow and noise characteristics of NACA0018 airfoil." *Journal of Wind Engineering and Industrial Aerodynamics* 95, no. 7 (2007): 511-531. https://doi.org/10.1016/j.jweia.2006.11.002

[15] Raghunathan, S., and O. O. Ombaka. "A thick symmetrical aerofoil oscillating about zero incidence angle." *International Journal of Heat and Fluid Flow* 7, no. 2 (1986): 155-159. https://doi.org/10.1016/0142-727X(86)90065-2

[16] Kharulaman, Liyana, Abdul Aabid, Fharukh Ahmed, Ghasi Mehaboobali, and Sher Afghan Khan. “Research on Flows for NACA 2412 Airfoil Using Computational Fluid Dynamics Method.” *International Journal of Engineering and Advanced Technology* 9, no. 1 (2019): 5450–5456. https://doi.org/10.35940/ijeat.A3085.109119

[17] Sajali, Muhammad Fahmi Mohd, Syed Ashfaq, Abdul Aabid, and Sher Afghan Khan. “Simulation of Effect of Various Distances between Front and Rear Body on Drag of a Non-Circular Cylinder.” *Journal of Advanced Research in Fluid Mechanics and Thermal Sciences* 62, no. 1 (2019): 53–65.

[18] Sajali, Muhammad Fahmi Mohd, Abdul Aabid, Sher Afghan Khan, Fharukh Ahmed Ghasi Mehaboobali, and Erwin Sulaeman. "Numerical investigation of flow field of a non-circular cylinder." *CFD Letters* 11, no. 5 (2019): 37-49.

[19] Aabid, Abdul, Azmil Afifi, Fharukh Ahmed Ghasi Mehboob Ali, Mohammad Nishat Akhtar, and Sher Afghan Khan. “CFD Analysis of Splitter Plate on Bluff Body.” *CFD Letters* 11, no. 11 (2019): 25–38.

[20] Khan, Sher Afghan, Abdul Aabid, Imran Mokashi, Abdulrahman Abdullah Al-Robaian, and Ali Sulaiman Alsagri. “Optimization of Two-Dimensional Wedge Flow Field at Supersonic Mach Number.” *CFD Letters* 11, no. 5 (2019): 80–97.

[21] Khan, Sher Afghan, Abdul Aabid, and Ahmed Saleel C. “CFD Simulation with Analytical and Theoretical Validation of Different Flow Parameters for the Wedge at Supersonic Mach Number.” *International Journal of Mechanical & Mechatronics Engineering IJMME-IJENS* 19, no. 1 (2019): 170–177.

[22] Aabid, Abdul, Zakir Ilahi Chaudhary, and Sher Afghan Khan. “Modelling and Analysis of Convergent Divergent Nozzle with Sudden Expansion Duct Using Finite Element Method.” *Journal of Advanced Research in Fluid Mechanics and Thermal Sciences* 63, no. 1 (2019): 34–51.

[23] Khan, Sher Afghan, Abdul Aabid, Fharukh Ahmed Mehaboobali Ghasi, Abdulrahman Abdullah Al-Robaian, and Ali Sulaiman Alsagri. "Analysis of area ratio in a CD nozzle with suddenly expanded duct using CFD method." *CFD Letters* 11, no. 5 (2019): 61-71.

[24] Fharukh, Ahmed GM, Abdulrehman A. Alrobaian, Abdul Aabid, and Sher Afghan Khan. "Numerical analysis of convergent-divergent nozzle using finite element method." *International Journal of Mechanical and Production Engineering Research and Development* 8, no. 6 (2018): 373-382. https://doi.org/10.24247/ijmperddec201842

[25] Khan, Sher Afghan, Abdul Aabid, and Maughal Ahmed Ali Baig. "CFD analysis of CD nozzle and effect of nozzle pressure ratio on pressure and velocity for suddenly expanded flows." *International Journal of Mechanical and Production Engineering Research and Development* 8, no. 3 (2018): 1147-1158. https://doi.org/10.24247/ijmperdjun2018119

[26] Aabid, Abdul, Ambreen Khan, Nurul Musfirah Mazlan, Mohd Azmi Ismail, Mohammad Nishat Akhtar, and Sher Afghan Khan. "Numerical simulation of suddenly expanded flow at Mach 2.2." *International Journal of Engineering and Advanced Technology* 8, no. 3 (2019): 457-462.

[27] Akhtar, Mohammad Nishat, Elmi Abu Bakar, Abdul Aabid, and Sher Afghan Khan. "Numerical Simulations of a CD Nozzle and the Influence of the Duct Length." *International Journal of Innovative Technology and Exploring Engineering (IJITEE)* 8 (2019): 622-630. https://doi.org/10.35940/ijitee.I1127.0789S219