JOURNAL OF ADVANCED MECHANICAL ENGINEERING APPLICATIONS
VOL. 1 NO. 1 (2020) 9-19

© Universiti Tun Hussein Onn Malaysia Publisher's Office

JAMEA

http://publisher.uthm.edu.my/jamea/index.php/jamea
e-ISSN : XXXX-XXXX

Journal of Advanced Mechanical Engineering Applications

Comparative Study on Several Type of Turbulence Model Available in ANSYS-Fluent Software for ONERA M6 Wing Aerodynamic Analysis

Mohd Zarif Md Shah[1], Bambang Basuno[1*], Aslam Abdullah[1]

[1]Faculty of Mechanical and Manufacturing Engineering,
Universiti Tun Hussein Onn Malaysia (UTHM), Parit Raja, Batu Pahat, 86400 Johor, MALAYSIA

*Corresponding Author

DOI: https://doi.org/10.30880/jamea.2020.01.01.002
Received 25th October 2020; Accepted 10th February 2020; Available online 29th February 2020

Abstract: ONERA M6 wing model is definitive computational fluid dynamic (CFD) validation case for aerodynamic investigations. Therefore, such investigation on aerodynamic characteristics is conducted with commercial CFD software known as ANSYS-Fluent software. The advantages of this software beside offer various flow solvers, this software also provides a various type of Turbulence Models can be implemented. In the present works, an investigation on capabilities of turbulence models available in the ANSYS-Fluent software based on Boussinesq hypothesis had been studied. The investigation had used five type Turbulence Models for evaluating the aerodynamic characteristics of the ONERA M6 wing model. These five turbulent models are: (1) Spalart-Allmaras, (2) k-ε Standard, (3) k-ε Realizable, (4) k-ω Standard and (5) k-ω SST turbulence models. The flow analysis are carried at two different angle of attacks namely at $\alpha = 3.06^0$ and $\alpha = 6.06^0$. These two angle of attacks correspond with Mach number $M_\infty = 0.84$ and the Reynolds number Re = 11.76×10^6. The comparison between ANSYS software with the experimental result as provided by AGARD AR-138 in term pressure coefficient distribution at several wing span location and overall aerodynamics characteristics in term lift coefficients shows that among those five turbulence models had been found that Spalart-Allmaras and k-ω SST turbulence model gives result close to the experimental results and the smallest number of iteration for getting a converge solution.

Keywords: Computational Fluid Dynamics, CFD, ANSYS-Fluent, Turbulence Model, Spalart-Allmaras, k-ε Standard, k-ε Realizable, k-ω Standard, k-ω SST

1. Introduction

Since 1960, the aerospace industry has revolutionized by integrating the Computational Fluid Dynamics (CFD) technique as part of its activities in both design & manufacturing as well as research & development. These technique gives engineers and scientists a robust data and saves cost in term of money and time compared to the orthodox technique, i.e. experiment method. The emerging of CFD technique delivers an enormous impact and transforms how computational mechanic, optimization and the associated design are done. Most of the cases involving the system behavior (e.g. a system that can't be simply calculated by conventional computation), the complexity of the geometry, the high cost of time and money, and the issue of material and space availability can be solved by using CFD technique. This is strongly supported by the proliferation of high-speed computers, giving the resolution and cell size of the CFD model domain to improve dramatically over the past few decades. To approve the claim, Airflow Sciences Corporation which has used both experimental and modeling methods since 1975, has made numerous comparisons between CFD

*Corresponding author: mohdzarif@outlook.com
2020 UTHM Publisher. All rights reserved.

modeling, physical modeling, and field testing. Results indicate that both types of models share the same accuracy when it comes to velocities and pressures [1].

The awareness of the importance of the CFD technique makes researchers and engineers find a better and versatile tool for problems they encounter. One of the reliable CFD tools is ANSYS-Fluent software which belongs to a commercial software company, ANSYS Incorporated. The tool represents a commercialized general-purpose CFD software package which is widely used by many people worldwide, ranging from academicians to industrial communities.

The present work uses ANSYS-Fluent version 16.1 with the purpose of evaluating the software capabilities in view of its turbulence modeling. Therefore, the focus of this paper is to investigate the capabilities and robustness of several turbulence models in ANSYS-Fluent software for flow past swept-wing model with a moderate angle of attack. It had been identified that the ANSYS-Fluent software provides various types of turbulence models that can be used in the flow analysis. In this respect, we use (1) Spalart-Allmaras, (2) k-ε Standard, (3) k-ε Realizable, (4) k-ω Standard and (5) k-ω SST turbulence models. These five types of turbulence models are applied in flow analysis over the wing model as provided by AGARD AR-138. This wing model is called the ONERA M6 wing, built and tested in a wind tunnel by Schmitt, V & F. Charpin in 1979 [2]. The models are tested at several flow conditions, namely at the angle of attack, α = 3.06 and α = 6.06, flow Mach number, M_∞= 0.84 and the Reynolds number, Re = 11.76×10^6.

The comparative results between these five turbulence models applied to the cases of relatively low angle of attack, α = 3.06° and medium angle of attack, α = 6.06°, indicate that the models' predictions are in close agreement with the experimental results. Detailed comparisons in terms of pressure coefficient distribution, C_p for different wing section and lift coefficient, C_L of the wing model will be presented further in this paper.

2. ANSYS-Fluent Software as a Tool in Wing Aerodynamic Analysis

Basically, ANSYS software is not a single purpose software, but represents multi-purposes software. This software can be used for solving various engineering problems ranging from fluid flow to mechanical structure analysis. The ANSYS fraction which is designed to deal with a fluid dynamics problem is called ANSYS-Fluent software.

In this present work, ANSYS-Fluent version 16.1 is used. Historically, Fluent is a CFD software as a result of collaboration work between Sheffield University and Creare Incorporated with the first version of Fluent software that was launched in October 1983. In May 2006, Fluent Incorporated was acquired by ANSYS Incorporated and become ANSYS-Fluent as known today [3].

This software solves the flow problems by using the generalized Navier-Stokes equations as its governing equations of fluid motion and by using the finite volume method as the manner the spatial discretizations are carried out. As ANSYS acts as a flow solver using a finite volume method, it provides various models of grid element in a flow domain such as hexahedral, polyhedral, prismatic, and tetrahedral mesh. In line with the governing equations of fluid motion in the form of Reynolds Averaged Navier-Stokes (RANS) equations, a turbulence model is required. RANS equations govern the transport of the averaged flow quantities, with the whole range of the scales of turbulence being modeled. There aretwo approaches in solving RANS equations namely the Boussinesq approach and Reynolds Stress Transport Models. However, the present work uses Boussinesq approach since it is a relatively low computational cost associated with the computation of the turbulent viscosity, μ_t. A comprehensive closure models based on Boussinesq approach are available in ANSYS-Fluent software, including Spalart-Almaras, k-ε and its variants, as well as k-ω and its variants.

In general, RANS equations describe the mathematical forms of continuity and momentum principles, and take a time (or ensemble) average (and drop the overbar on the mean velocity, $\bar{u}$) which finally yields the ensemble-averaged momentum equations. They can be written in a Cartesian tensor form as

$$\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x_i}(\rho u_i) = 0 \qquad (1)$$

$$\frac{\partial}{\partial t}(\rho u_i) + \frac{\partial}{\partial x_j}(\rho u_i u_j) = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j}\left[\mu\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} - \frac{2}{3}\delta_{ij}\frac{\partial u_l}{\partial x_l}\right)\right] + \frac{\partial}{\partial x_j}\left(-\rho\overline{u_i' u_j'}\right) \qquad (2)$$

Basically, equation (1) and (2) is called RANS equations and have the same general form as the Navier-Stokes equations, with the velocities and other solution variables now representing ensemble-averaged (or time-averaged) values with an additional term representing the effects of turbulence, $-\rho\overline{u_i' u_j'}$. The additional terms of turbulence, $-\rho\overline{u_i' u_j'}$, are called Reynold Stresses, which must be modelled in order to close the equations (2). In present work, therefore, the Boussinesq hypothesis is used to relate the Reynolds stresses to the mean velocity gradients [4], [5]:

$$-\rho\overline{u_i' u_j'} = \mu_t\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right) - \frac{2}{3}\left(\rho k + \mu_t\frac{\partial u_k}{\partial x_k}\right) \qquad (3)$$

3. The ONERA M6 Wing Model

The present work uses a wing model adopted from AGARD AR-138 [2] with the wing geometry data is given in Table 1 as follows:

Table 1 - Wing Geometry of ONERA M6 wing

Wing Semi Span	1.1963 meter
Mean aerodynamic center	0.64607 meter
Aspect ratio	3.8
Taper ratio	0.56
Leading-edge sweep angle	30°
Trailing edge sweep angle	15.8°
Sweep angle at 25% chord	26.7°

The wing uses a uniform shape cross-section airfoil is known as asymmetrical airfoil of ONERA D section. The pressure measurements are carried out in a spanwise direction over seven different stations (non-dimensional longitudinal station in percentage) as shown in Figure 2. The location of these seven stations is given in Table 2 with 34 pressure tap for each station. The locations of pressure tap as shown in Figure 1.

Table 2 - Location of pressure tap along the wing semi-span

Section	Relative Spanwise Position in percentage (η)	Spanwise Position (meter)
S1	0.2	0.23926
S2	0.44	0.526372
S3	0.65	0.777595
S4	0.8	0.95704
S5	0.9	1.07667
S6	0.95	1.136485
S7	0.99	1.184337

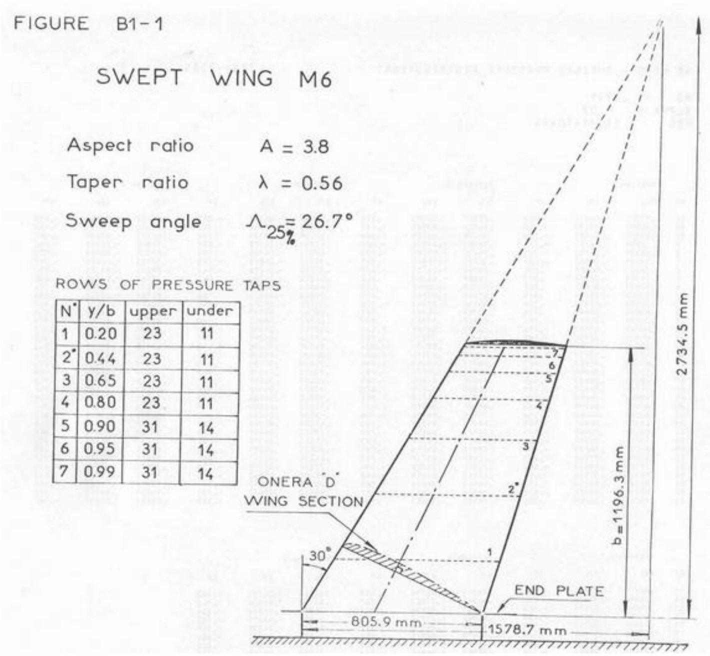

Fig. 1 - Layout of ONERA M6 Wing Planform [2]

4. Methodology

In the manner of finding solutions on how to solve the flow problem by using ANSYS-Fluent Software, it can be divided into the five main steps, namely (1) Geometry definition, (2) Meshing Flow domain, (3) Solution Setup, (4) Solution Methods & Calculation Tasks, and (5) Result.

4.1 Geometry Definition

The geometry of the object immersed in the flow field needs to be defined accurately. Basically in defining the geometry of the model, the work can be carried out by using any of CAD software. However, the present works use SOLIDWORK software. In defining wing geometry, the data wing planform use is given in Table 1. However for the airfoil section, the present work uses the modified airfoil coordinate data as provided by NPARC Alliance [6]. The modification airfoil data is required since the original data generate non zero thickness at the trailing edge.

The geometry data generated by SOLIDWORK software, then the model is transferred to ANSYS-DesignModeller. Figure 2 shows the shape of the wing planform seen through ANSYS-DesignModeller. Line A and Line B, that appear in Figure 2 indicate a splitting of wing surface for a grid refinement used later in the meshing process. While Figure 3, shows the setting of the boundary condition will be applied over the flow domain in which in the spanwise cross-section direction has a C-topology and in streamwise cross-section has O-topology.

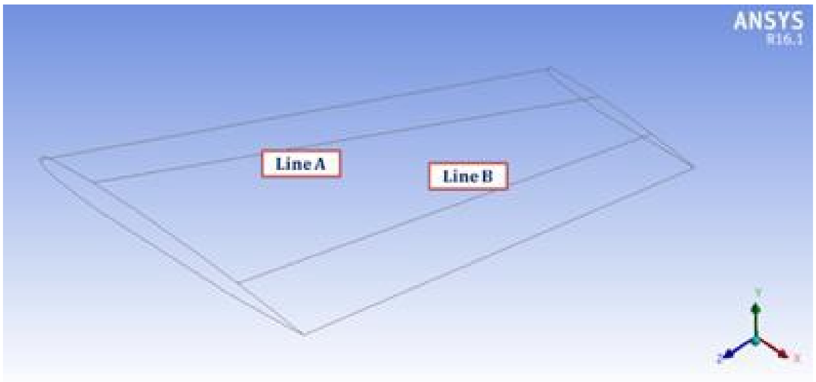

Fig. 2 - ONERA M6 wing geometry

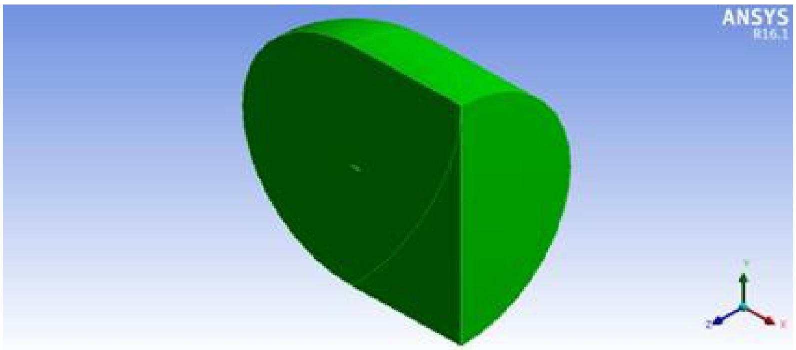

Fig. 2 - Pressure far-field boundary domain

4.2 Meshing and Designation of Far-Field Boundary Condition

In the computational simulation, the meshing process plays a vital role in determining the numerical solution to converge closely to actual condition or not. Here the unstructured mesh is used in defining the mesh flow domain. Figure 4 shows how the grid and the boundary conditions are applied in solving this flow problem. Here, there are five boundary conditions that had been implemented, namely the inlet, outlet, wing surface, wingtip, near side and far side boundary condition.

Figure 5 shows the close up of the grid distribution close to the wing surface in which in meshing flow domain had been included the boundary layer thickness. For the ONERA M6 wing model, the boundary layer thickness is defined by setting the non-dimensional value of y^+ is equal to 1. Setting such value makes the viscous sub-layer is included in the near body surface [7]. This approach gives the first-layer thickness of the boundary layer which is equal to 4.81 x 10^{-5} meter with a growth rate of 1.2

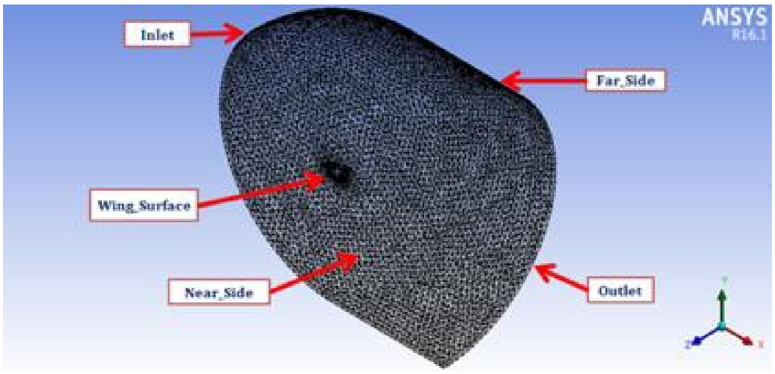

Fig. 4 - Named pressure far-field boundary condition

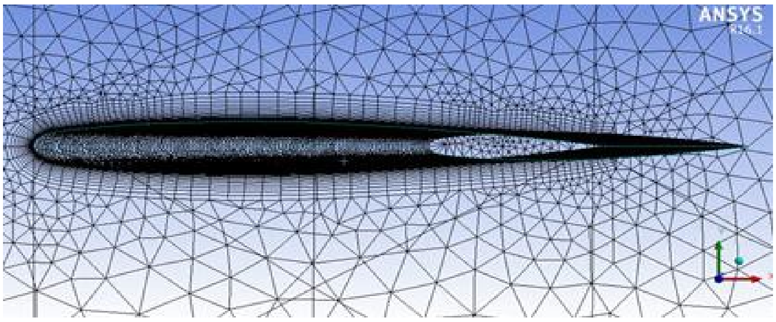

Fig. 5 - Boundary layer of ONERA M6 Wing

4.3 ANSYS-Fluent Solution Setup

As the meshing flow domain has been set up, however, there are four other steps need to be done. These four steps are, (1) Solver Type Determination, (2) Turbulent Model Selection, (3) Fluid Material and Boundary Condition Definition, and (4) Defining the Reference Values.

In determining the type of solver, the ANSYS-Fluent software provides 5 types of solvers, they are namely:

- Pressure based solver + Coupled algorithm
- Pressure based solver + SIMPLE (Semi-Implicit Method for Pressure-Linked Equation) algorithm
- Pressure based solver + SIMPLEC (Semi-Implicit Method for Pressure Linked Equations-Consistent) algorithm
- Pressure-based solver + PISO (Pressure-Implicit with Splitting of Operators) algorithm
- Density-based coupled solver.

The present work used a pressure-based solver + Coupled algorithm. The reason to use the pressure-based solver + Coupled algorithm is more rapid and monotonic convergence rate and hence faster solution times since the algorithm solved the continuity and momentum equation in coupled fashion thus eliminating the approximation produced by segregated solution approach where the momentum and continuity equations are solved separately. Therefore, by eliminating the approximations due to isolating the equations permits the dependence of the momentum and continuity on each other. On top of that, the algorithm used improved the robustness of the solution such that errors associated with initial conditions, nonlinearities in the physical models, and stretched and skewed meshes do not affect the stability of the iterative solution process [8].

In view of the turbulent flow model, ANSYS-Fluent software provides nine main turbulent models. Some of the methods use multiple turbulent models such as the k-ε models, the k-ω models, and the Reynolds stress models. These nine turbulent models belong to the class of either one-, two-, three- or four-equation turbulent models. The Spalart-Allmaras turbulent model which represents the one-equation turbulent model used a Boussinesq hypothesis in solving a transport equation for the kinematic eddy turbulent viscosity parameter, $\tilde{\nu}$. While the turbulent model called k-ε Standard, k-ε Realizable, k-ω Standard and k-ω SST represent a two-equation model. Details of various turbulent models can be obtained in [9-14]

In setting the fluid material and boundary condition to the case of flow past through the ONERA M6 wing, the third step of the ANSYS-Fluent setup is presented in Table 3. For the purpose of post-processing, the reference values in step four need to be defined as shown in Figure 6. The user can control the reference values that are used in the computation of derived physical quantities and non-dimensional coefficients

Table 3 - Type and setting value at the Boundary Condition

Boundary	Type	Condition	Viscosity (kg/m-s)
- Inlet - Outlet - Far-Side	Pressure Far-Field	-Temperature = 300K - Mach Number = 0.84 - Pressure = 14.7 kPa	1.846e-05
- Near_Side	Symmetry	Atmospheric Pressure = 0	1.846e-05
- Wing_Surface	Wall	Atmospheric Pressure = 0	1.846e-05
- Wing_Surface			

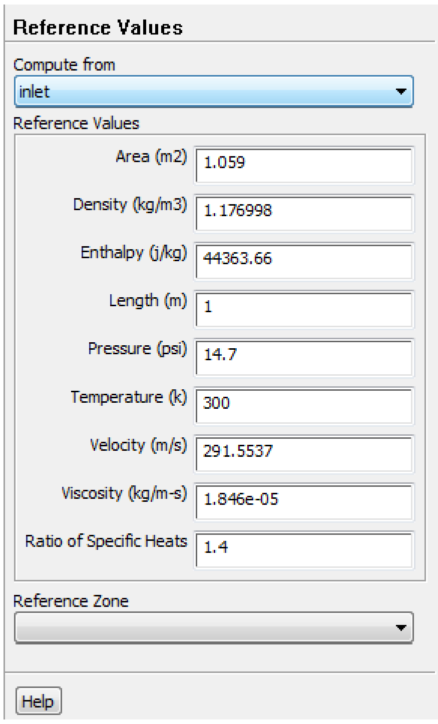

Fig. 6 - Reference values setting of the wing model

4.4 ANSYS-Fluent Solution Method & Calculation Tasks

In this stage, there are four elementary steps that need to be carried out. They are namely: (1) Solution Method, (2) Monitors, (3) Solution Initialization and (4) Run Calculation.

In the solution method in which the pressure-based solver + Coupled scheme is used, the spatial discretization input needs to be defined as given in Figure 7 (a). For achieving a convergent solution, the required setting value in the residual monitor as shown in Figure 7 (b) in which the setting value for the continuity and velocity are set in order of magnitude 0.001.

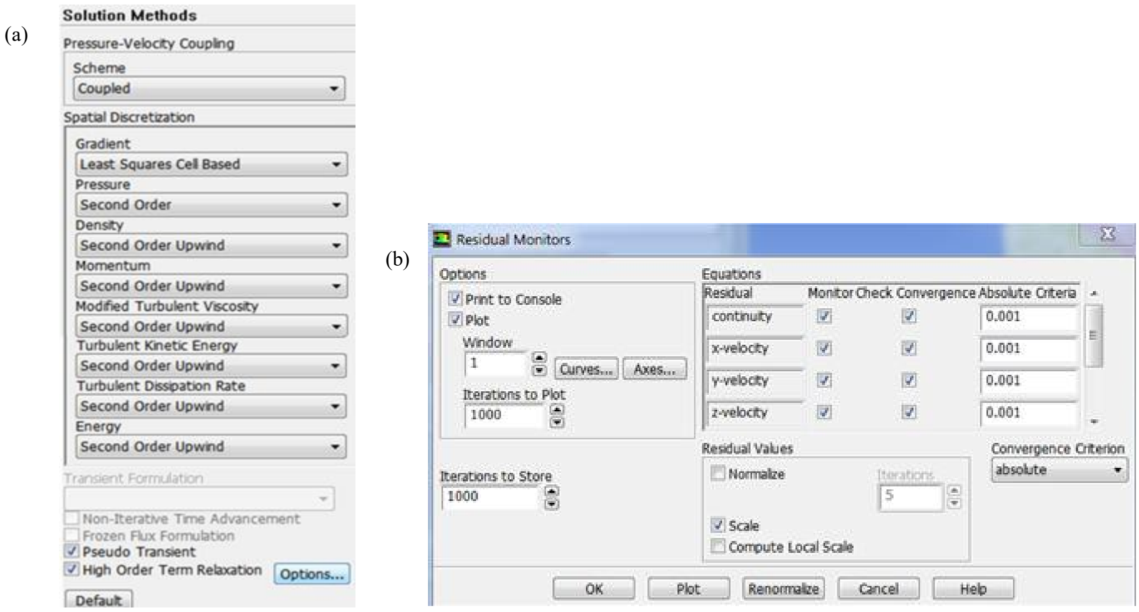

Figure 7 - (a) Solution Methods for Pressure-Velocity Coupling and Spatial Discretization settings; (b) Residual Monitors

It is necessary to be noted here, by definition, the residuals are the error of magnitudes for equations as iterations progress. In theory, the residual should reach zero as the solution converges, however, in an actual calculation, the residuals will decay to some small value ("round-off") and then stop changing ("level out"). In this present work, the Scaled Residual monitor is based on FLUENT default in which the solution will converge up to 10^{-3} for all equation except for energy in which the criterion is 10^{-6} [4].

For the next step, a Solution Initialization is needed for the solver to start with its first iteration process. The present work uses Hybrid Initialization method to provide the initial value for the flow variables in every grid cell. The ANSYS-Fluent will carry out initialization flow variables for the velocity and pressure field by solving a Laplace equation with the setting the boundary condition follows as the case of External-Aero problem.

After the physical setup and the Solution Control are completed, the calculation process can be carried out. The calculation for the different turbulent model can be easily carried out just by merely changing the part of the turbulent model setting. The result of different turbulent models is presented in the following subchapter.

5. Results & Discussions

As the geometries, meshing and solution method has been set up, the result of the simulation is performed in ANSYS- Fluent post-processor. In this present work, two experimental test cases from AGARD [2], test case number 2308 and 2565 are used for a comparison purpose. These two test cases represent the wind tunnel test over ONERA M6 wing conducted at the Reynolds number, *Re*, Mach number, M_∞ and the angle of attack, α as shown in Table 4

Table 4 - Experimental Flow Condition

Test Number	Reynolds Number, (*Re*)	Mach Number, (M_∞)	Angle of Attack, α (degree)
2308	11.76×10^6	0.84	3.06^0
2565	11.76×10^6	0.84	6.06^0

The comparison result of pressure coefficient distribution, C_p for different spanwise locations between ANSYS-Fluent simulation and experimental result for the test case number 2308 is shown in Figure 8. While Figure 9 shows the ANSYS-Fluent simulation comparison to the test case number 2565.

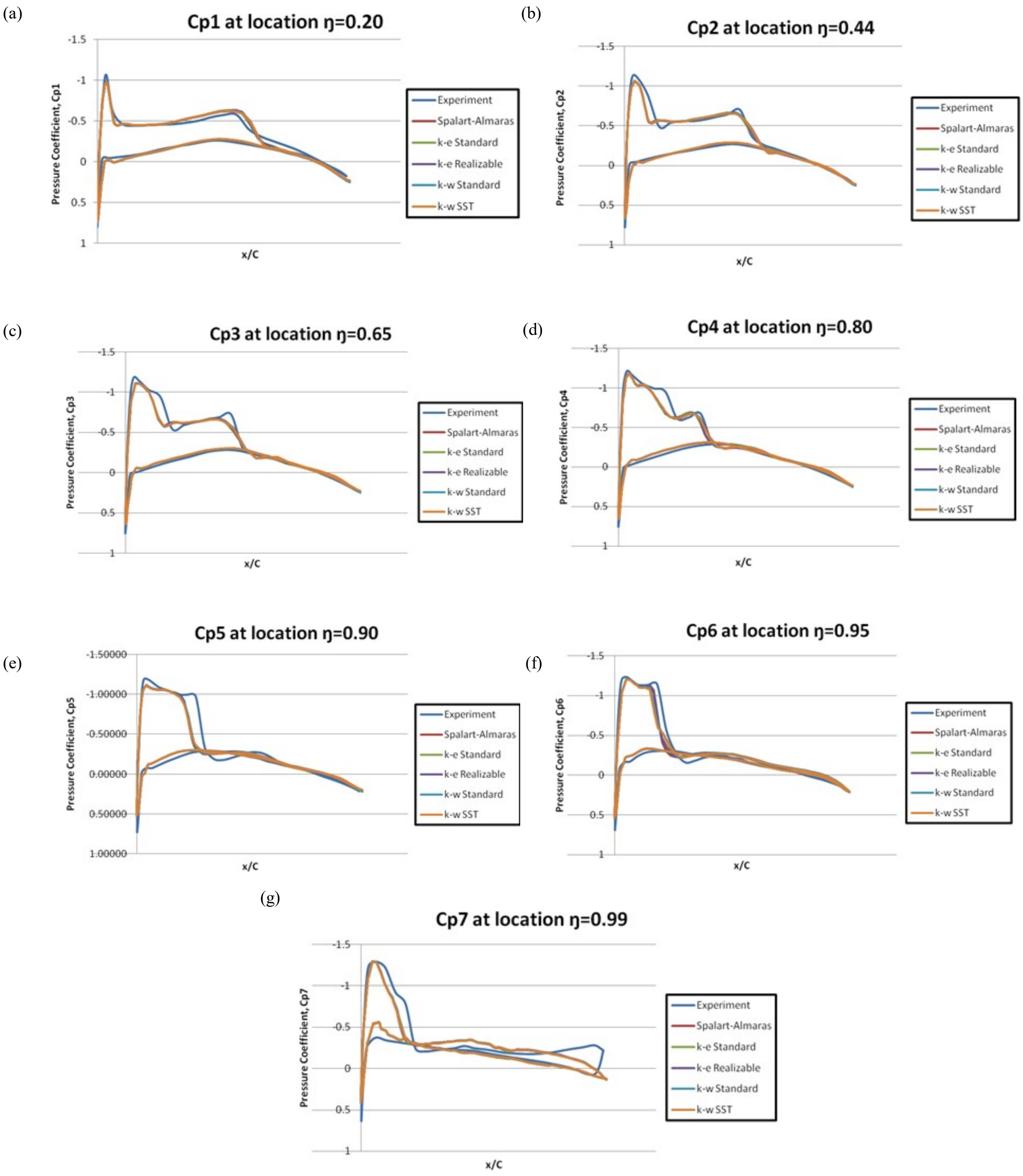

Figure 8 - Comparison result of ONERA M6 wing pressure coefficient distribution, C_p along the chord at seven spanwise stations between experimental result and ANSYS-Fluent simulation with five types of turbulence model for the angle of attack, α = 3.06

Figure 8 shows the comparison result at the angle of attack α = 3.06^0 indicates all turbulent models in used are able to produce the result as provided by the experimental result. However if the angle of attack is increased to α = 6.06^0, their comparison result as shown in Figure 9. These figures indicate that each turbulent model has its own solution. As a result, the pressure coefficient distribution between one turbulent model differs from other turbulent models. Their difference becomes apparent as the pressure coefficient distribution toward the wingtip, especially from the station ŋ = 0.65 to ŋ = 0.99.

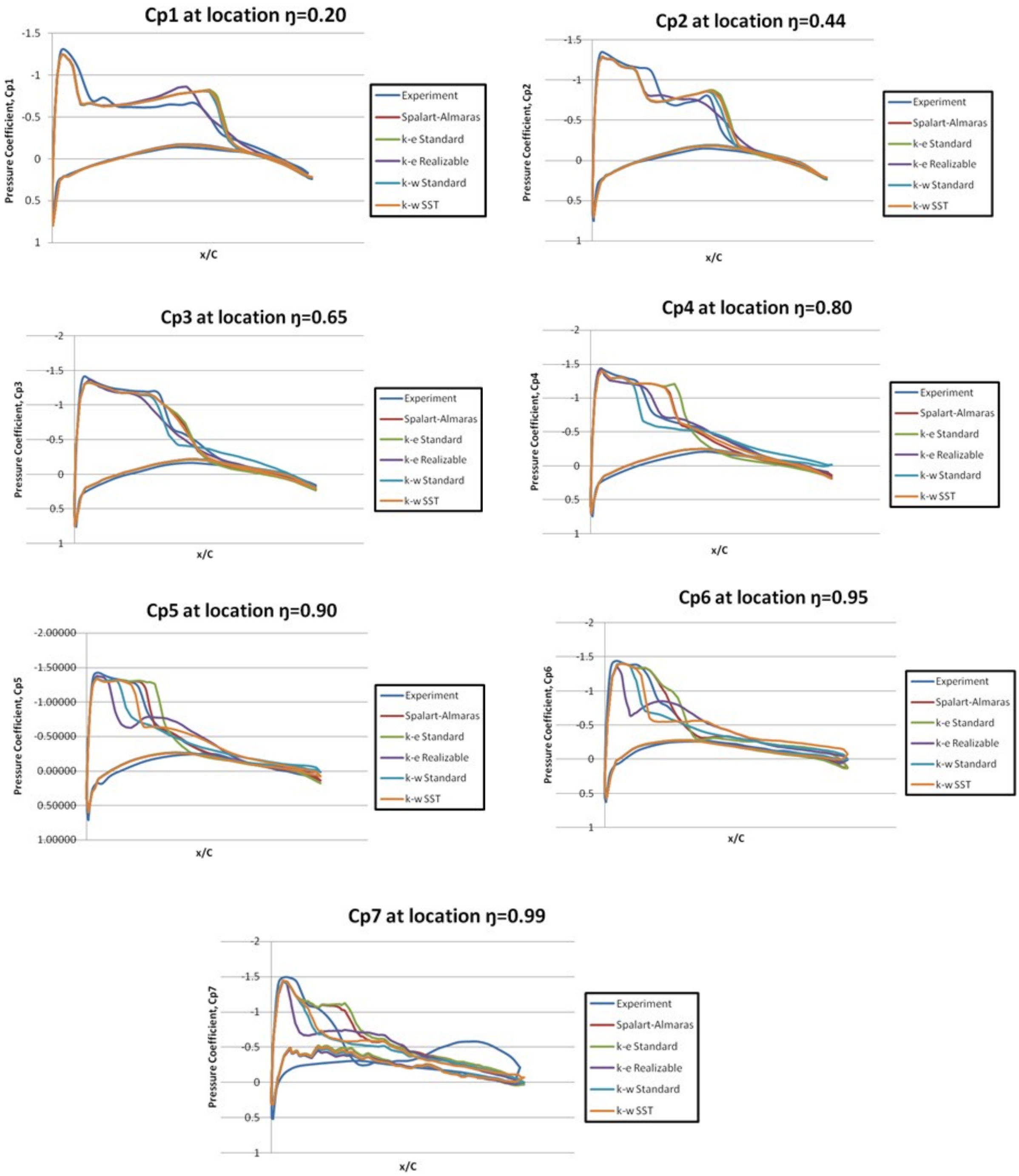

Fig. 9 - Comparison result of ONERA M6 wing pressure distribution, C_p along the chord at seven spanwise stations between experimental result and ANSYS-Fluent simulation with five types of turbulence model for the angle of attack, α = 6.06°

The comparison results in term of lift coefficient, C_L for five types of turbulence model is shown in Table 5. The lift coefficient was obtained by the integration of the pressure coefficient over the wing surface at the angle of attack, $\alpha=3.06^0$ and $\alpha = 6.06^0$. In term of the lift coefficient, C_L all turbulence models produce their different results no more than 5% compared to the experiment results. This result indicates that the ANSYS-Fluent software represents the software which able to produce the result as the experiment as far as the angle of attack relatively moderate with the typical wing planform as ONERA wing M6 model.

Table 5 - Absolute Relative Errors (in percentage, %) of Lift Coefficient, C_L for five types of turbulence model in test case 2308 and 2565

Turbulence Model	Test Case 2308 - Angle of Attack = 3.06^0			Solution Iteration Step
	C_L ANSYS-Fluent Simulation	C_L Experiment	C_L of Absolute Relative Error (%)	
Spalart-Allmaras	0.248292225	0.258334297	3.887239323	60
k-ε Standard	0.249059457	0.258334297	3.590247284	78
k-ε Realizable	0.248502121	0.258334297	3.805989627	212
k-ω Standard	0.248529558	0.258334297	3.795368678	60
k-ω SST	0.245732189	0.258334297	4.878217225	60
Turbulence Model	Test Case 2565 - Angle of Attack = 6.06^0			Solution Iteration Step
	C_L ANSYS-Fluent Simulation	C_L Experiment	C_L of Absolute Relative Error (%)	
Spalart-Allmaras	0.501181745	0.490815571	2.112030246	62
k-ε Standard	0.504585846	0.490815571	2.805590311	74
k-ε Realizable	0.478383404	0.490815571	2.532961009	357
k-ω Standard	0.482412254	0.490815571	1.71211304	113
k-ω SST	0.502049532	0.490815571	2.288835429	62

However, in term of the required number of iteration to reach convergence solution the Spalart-Allmaras and k-ω SST turbulence models have the same order of magnitude and the smallest one. From the point of view pressure distribution $\boldsymbol{C_p}$, the lift coefficient and the required number of iteration, it can be said that the Spalart-Allmaras and k-ω SST turbulence models may represent the most suitable turbulent model to be used for solving a flow problem past through wing belong to the class ONERA M6 wing model.

5. Conclusion

Since the overall behavior for all five turbulence models for both test cases (test case 2308 and test case 2565) with the angle of attack, $\alpha = 3.06^0$ and $\alpha = 6.06^0$ are in reasonably good agreement with the experimental result. Based on pressure coefficient distribution, C_p for both angle of attack, all the turbulence models predict the shock formulation on the upper surface of the ONERA M6 wing without any significant differences from the experiment. This can be seen after integrating the pressure coefficient, C_p in order to obtain the lift coefficient, C_L. The lift coefficient is then compared between ANSYS-Fluent simulation and experimental result for absolute relative error. Based on table 5, lift coefficient, C_L of relative absolute error for both angle of attacks shows a good agreement with the experimental result which is below 5%. As a conclusion, ANSYS-Fluent software with turbulence model of (1) Spalart-Allmaras, (2) k-ε Standard, (3) k-ε Realizable, (4) k-ω Standard and (5) k-ω SST turbulence models are robust and adequate for use in determining the aerodynamic problems and fluid flow past a typical wing model with medium angle of attack. However, among five turbulence models, Spalart-Allmaras and k-ω SST turbulence model show a good performance in robustness, accuracy and less time-consuming.

Further studies in various wing designs such as the straight wing, delta wing, cropped delta wing, the elliptical wing should be performed to understand the flow pattern, aerodynamic characteristics and the cause for the differences between ANSYS-Fluent turbulence modeling and experimental data. The capability of turbulence models available in ANSYS- Fluent software can be investigated and compared with the various wing design. In particular, the user should consider the benefits to integrate multiple turbulence models of the same class such as used in the Reynolds Stress Model.

Acknowledgement

The authors would like to thank Faculty of Mechanical and Manufacturing Engineering Universiti Tun Hussein Onn Malaysia (UTHM), Parit Raja, Batu Pahat, 86400 Johor, MALAYSIA

References

[1] Kevin W. Linfield & Robert G. Mudry, "Pros and Cons of CFD and Physical Flow Modeling: A White Paper. Livonia, Michigan (USA): Airflow Science Corporation. 2008.

[2] Schmitt, V. and F. Charpin, "Pressure Distributions on the ONERA-M6-Wing at Transonic Mach Numbers," *Experimental Data Base for Computer Program Assessment.* Report of the Fluid Dynamics Panel Working Group 04, AGARD AR 138, May 1979

[3] Center for Computational Science (2018). "*A Brief History of Fluent*." Internet: https://www.ccs.uky.edu/UserSupport/SoftwareResources/Fluent/, retrieved on November 11, 2018

[4] ANSYS Fluent Version 16.1. *ANSYS Fluent Theory Guide*. 2015.

[5] Hinze. J. O. *Turbulence.* McGraw-Hill Publishing Co., New York, 1975

[6] National Aeronautics & Space Administrations (NASA) (2012). *NPARC Alliance Verification & Validation Archive*. Retrieved on September 7, 2018, from https://www.grc.nasa.gov/www/wind/valid/archive.html

[7] Schlichting, H. & Gersten, K. *Boundary Layer Theory*. 9th Edition. Berlin. Springer Nature. 2017

[8] Franklyn J. Kelecy. Coupling Momentum and Continuity Increases CFD Robustness in ANSYS Advantage Volume II Issue 2 2008. ANSYS, Inc. pp. 49-51.

[9] Spalart, P. and Allmaras S. A one-equation turbulence model for aerodynamic flows. Technical Report AIAA-92-0439, American Institute of Aeronautics and Astronautics, 1992

[10] Launder, B. E. and Spalding, D. B. The Numerical Computation of Turbulent Flows, Compute. Methods Appl. Mech. Eng., Vol. 3, pp. 269-289. 1974.

[11] T.-H. Shih, W. W. Liou, A. Shabbir, Z. Yang, and J. Zhu. "A New - Eddy-Viscosity Model for High Reynolds Number Turbulent Flows - Model Development and Validation". Computers Fluids. 24(3). 227-238. 1995

[12] D. C. Wilcox. Turbulence Modeling for CFD. 1st Edition. La Canada, California. DCW Industries, Inc. 1993, 1994

[13] F. R. Menter. "Two-Equation Eddy-Viscosity Turbulence Models for Engineering Applications". AIAA Journal. 32(8). 1598-1605. August 1994.

[14] F. R. Menter. "Review of the SST Turbulence Model Experience from an Industrial Perspective". International Journal of Computational Fluid Dynamics. Volume 23, Issue 4. 2009