Journal of Engineering and Technology

ISSN 2180-3811 eISSN 2289-814X https://journal.utem.edu.my/index.php/jet/index

THE APPLICATION OF ANSYS-FLUENT SOFTWARE FOR AERODYNAMIC ANALYSIS ON RECTANGULAR AND MODERATE SWEPT WING PLANFORM

M. Z. Md Shah*[1], B. Basuno[2], A. Abdullah[3] and M. F. Pairan[4]
[1234] Faculty of Mechanical and Manufacturing Engineering, Universiti Tun Hussein Onn Malaysia Johor, Malaysia.
*mohdzarif@outlook.com

Article history:
Received Date: 2021-01-28
Accepted Date: 2021-06-27

Keywords: ANSYS-Fluent software, Computational fluid dynamics, Moderate swept wing, Rectangular wing, Turbulence model

Abstract— ANSYS-Fluent software represents a CFD software having the capability for solving various engineering flow problems. Besides offering a variety of flow solvers, this software also offers various type of turbulence model can be used in the flow analysis. The present work focuses on the use of this software applied to two type wing models, a moderately swept wing and (2) a rectangular wing planform. The moderately swept wing geometry and experimental data were obtained from AGARD AR-138, whereas the rectangular wing planform was obtained from RTO-TR-026. The first model evaluated by using five different turbulent models, namely (1) Spalart-Allmaras, (2) k-ε Standard, (3) k-ε Realizable, (4) k-ω Standard and (5) k-ω

SST turbulence models. Comparisons result with AGARD shows that all turbulent models are able to provide in a good agreement. However, Spalart-Allmaras and k-ω SST turbulence models give less CPU time than the others. These two turbulent models then applied to the case of a rectangular wing plan form. The result from the second test case, the k-ω SST turbulence models, give a more accurate result compared to the Spalart-Allmaras turbulence models. It gives a better result compared with the Spalart-Allmaras turbulence models. The k-ω SST turbulence model makes the ANSYS-Fluent result just differ 11.5% from the experiment result while for the Spalart-Allmaras turbulence model differs 14.05%. Here it can be concluded that k-ω SST turbulence mode may represent a suitable turbulence model for solving flow over a rectangular wing to the moderate swept wing plan form.

I. Introduction

Computational Fluid Dynamic or known as CFD, is one of the available numerical approaches that can be used for solving flow problems. The importance of the CFD approach makes a researcher and engineers find a better, versatile, and friendly interface tools to the problems they encounter. One of the available CFD tools is ANSYS-Fluent software which belongs to commercialize software company, ANSYS Incorporated. This software represents a commercialized general-purpose CFD software package that is widely used by many people worldwide, ranging from academicians to industrial communities.

The ANSYS FLUENT software represents CFD software which already is designed to have various type of flow solver and turbulence models [1]. To test the reliability of the flow solver and turbulence models available in the software, such test case is introduced in order to evaluate the software as a tool to solve the flow problem. Such a test case is to solve the flow problem past a different wing planform with several types of turbulence models. It had been identified that the ANSYS–Fluent software provides various types of turbulence models that can be used in 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 turbulence models are used to evaluate the aerodynamics characteristics of a swept-wing model is called ONERA M6 Wing. The geometry and experimental results for this wing model are already available in AGARD AR-138 [2]. The wings models are tested at several flow conditions, namely, at the angle of attack, α = 3.06⁰ and α = 6.06⁰ in the flow with Mach number M_∞ = 0.84 and the Reynolds number Re = 11.76×10^6. Comparison result between these five turbulence models and the experiment results indicate that these five turbulent models are able to provide the solution which is good agreement with the experiment with their differences less than 5% to the experimental results. However, among these five turbulent models, the Spalart-Allmaras and *k-ω* SST turbulent model may represent the best turbulent model among them. These two models besides giving the smallest difference to the experimental result, they also make ANSYS-Fluent converged at less required iteration number [3]. The implementation of Spalart-Allmaras and *k-ω* SST turbulent model is applied to the case of flow past through a rectangular wing with supercritical airfoil cross-section.

The geometry and experimental data for this rectangular planform is provided by RTO-TR-026 [4], in which the experiment is conducted at the angle of attack, α = 2.00º, Mach number M_{∞} = 0.802 and the Reynolds number Re = 0.401×10^{7}. These two turbulent models in term of the lift coefficient C_L and pressure coefficients distributions along the chord line at the various spanwise direction are in good agreement to the experimental results.

The present paper is divided into five main sections, namely, Introduction, Geometry Data of Wing Planform, The Implementation of ANSYS-Fluent Software, Analysis of Result and Discussion and Conclusion. In the second section, the geometry data of swept wing and rectangular wing model is presented. In the third section, the steps of ANSYS-Fluent software implementation are shown and discussed. In the fourth section, the analysis and comparison results in terms of pressure coefficients distribution, C_p and lift coefficient C_L is discussed. In the last section, the conclusion and indications further investigations are discussed.

II. Geometry Data of Wing Planform

A. Swept Wing Model

This wing model is known as ONERA M6 Wing in the present work provided by AGARD AR-138 [2]. The wing geometry data is given in Table 1.

Table 1: Wing Geometry of ONERA M6 wing

Wing Semi Span	1.1963 m
Mean aerodynamic centre	0.64607 m
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 a symmetrical airfoil of the ONERA D section. The pressure coefficient measurement, which had been conducted to this wing model, was carried out over seven sections, as shown in Table 2. The data of wing geometry and the number of pressures taps at each section, as shown in Figure 2. While schematics locations of pressure tap are shown in Figure 1.

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

Section	Relative Spanwise Position in percentage (ŋ)	Spanwise Position (m)
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

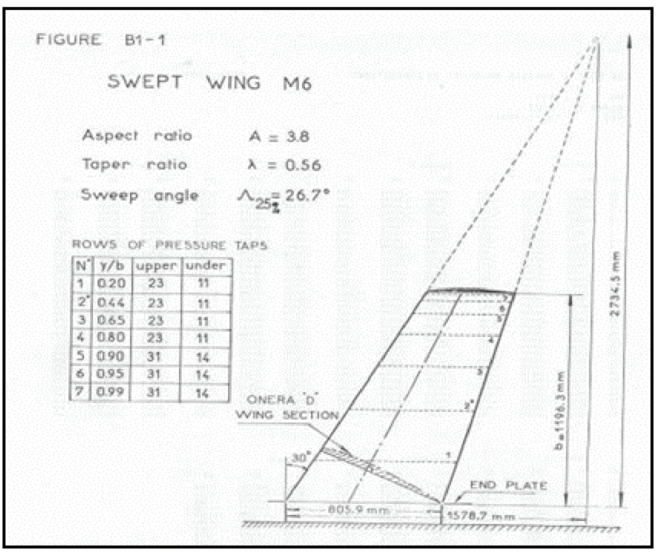


Figure 1: Layout of ONERA M6 Wing Planform [1]

B. Rectangular Wing Planform

The second wing model with supercritical airfoil cross-section used in the present work is adopted from RTO-TR-026 [4][5]. This wing is known as Rectangular Supercritical Wing, or RSW, with the wing geometry data is given in Table 3, and the location of the pressure tap along spanwise (non-dimensional longitudinal station in percentage) is shown in Table 4.

Table 3: Wing Geometry of RSW

Section	Relative Spanwise Position in percentage (ŋ)	Spanwise Position (m)
S1	0.309	0.3767
S2	0.588	0.7168
S3	0.809	0.9862
S4	0.951	1.1593

Table 4: Location of pressure tap of RSW semi-span

Wing Semi Span	1.219 m
Aspect Ratio	2
Taper Ratio	1.00
Leading & Trailing Swept	Unswept
Area of Planform	1.786 m^2
Wing Centreline Chord	0.6096 m

At each section, there are 29 pressure taps in which both upper and lower surfaces have 14 pressure taps, and one pressure tap is located at the nose of the wing leading edge. The position of these 29 pressures taps points are shown in Figure 2 and listed in Table 5. The airfoil used in this experiment is illustrated in Figure 3 as the value represents in inches. This airfoil is unsymmetrical and was derived by rationing the thickness of an 11% airfoil to 12% while keeping the same mean camber line. The trailing edge thickness is design to 0.7% chord by rotating the lower cusp area.

Table 5: Pressure Orifice Location and Type

x/c	Type
0.000	Tube to Transducer
0.003	Tube to Transducer
0.050	Tube to Transducer

0.100	Tube to Transducer
0.200	Tube to Transducer
0.260	In Situ
0.320	In Situ
0.380	In Situ
0.440	In Situ
0.500	In Situ
0.560	In Situ
0.620	In Situ
0.700	Tube to Transducer
0.800	Tube to Transducer
0.900	Tube to Transducer

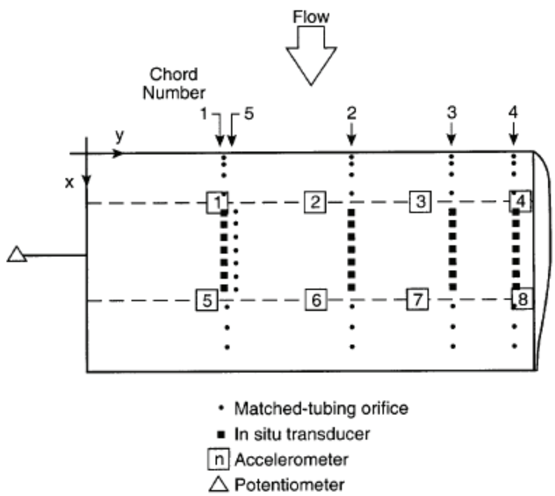


Figure 2: Instrumentation layout for the RSW model

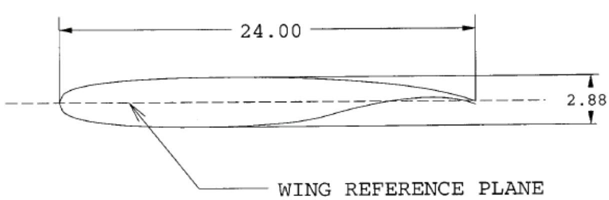


Figure 3: Airfoil for rectangular supercritical wing

III. The Implementation of ANSYS-Fluent Software

In the manner of how to predict the flow field pattern and flow characteristics by use of ANSYS-Fluent software, it can be divided into five main phases as shown in Figure 4 [6-8].

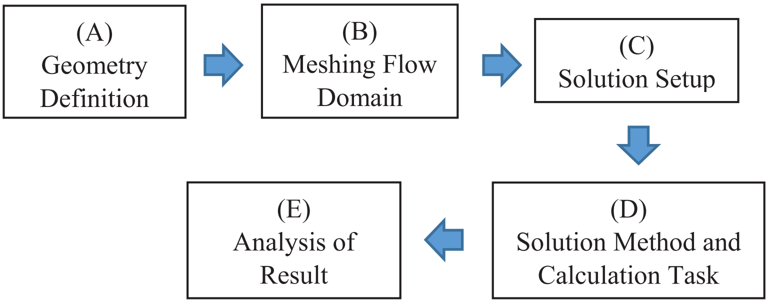
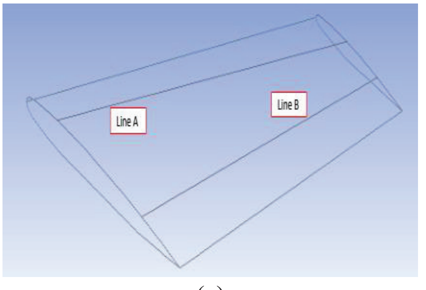


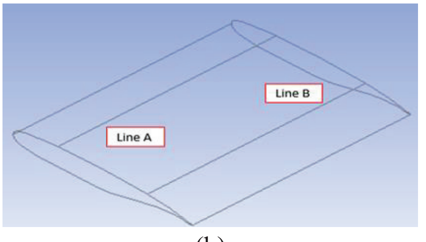
Figure 4: Main Phase of ANSYS-Fluent Software

A. Geometry Definition

The geometry immersed in the flow field needs to be accurately defined. Therefore, SOLIDWORK software is used in defining all the three wing models before the wing models transfer to the ANSYS-Design Modeller. In the case of a swept wing, a modified airfoil coordinate data is used, as discussed in [3]. For straight wing planform, the airfoil coordinate modification is not needed. Figure 5 shows the shape of the wing planform, as seen through ANSYS-Design Modeller. Line A and Line B that appear in Figure 5(a) and (b), indicate a splitting of wing surface for a grid refinement used later in the meshing process.



(a)



(b)

Figure 5: (a) ONERA M6 Wing (Swept Wing) and (b) Rectangular Wing Model Geometry

The setting of boundary condition is shown in Figure 6 that will be applied for both wing models in which in the spanwise cross-section direction has a C-topology and in streamwise cross-section has O-topology.

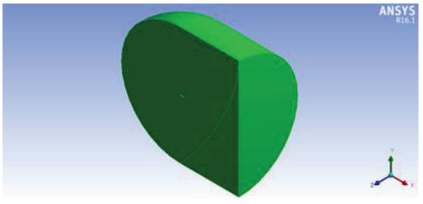
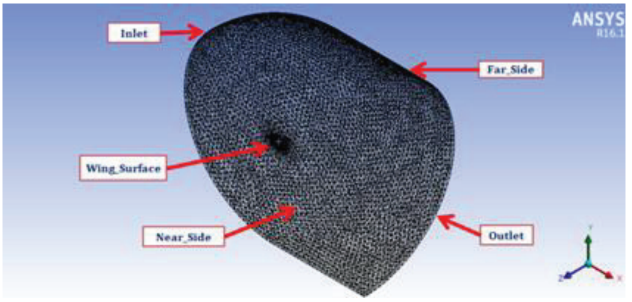


Figure 6: Pressure Far-Field Boundary Domain

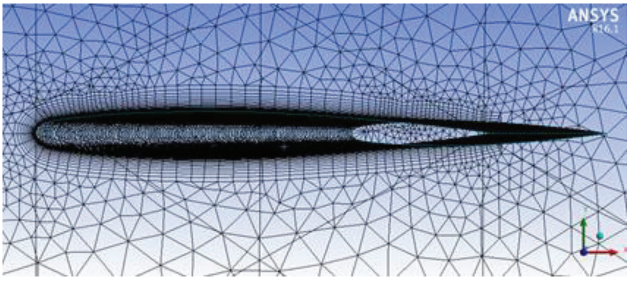
B. Meshing and Designation of Far-Field Domain

In CFD simulation, the meshing process plays important roles in determining the numerical solution converge close to the actual condition or not. Here, an unstructured mesh is used in determining the mesh flow domain. As shown in Figure 7(a), there are five distinctively boundary conditions that had been implemented for all two wing models, namely the inlet, outlet, wing surface, wingtip, near side and far side boundary condition. Figure 7(b) shows one of the examples of the close-up of the grid distribution close to the wing surface in which the meshing flow domain had been included the boundary layer thickness. In the present work, the boundary layer thickness is defined by setting the non-dimensional value of y^+ is equal to 1 is applied for all wing models. Setting such value makes the viscous sub-layer is included in the near body surface [9]. 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.

For an ONERA M6 wing, the number of nodes consists of 350942 nodes, and the number of elements consists of 913530 elements. While for the Rectangular wing models, the number of nodes represents 403776 nodes, and the number of elements is 967162 elements.



(a)



(b)

Figure 7: (a) Named Pressure Far-Field Boundary Condition; (b) Boundary Layer of ONERA M6 Wing

C. Solution Setup

Once the geometry definition and meshing flow domain have been set up, there are four steps that need to be done in the Solution Setup. 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 the context of flow solver, ANSYS-Fluent software provides several types of solver, as discussed in [3]. The present work used a pressure-based solver + Coupled algorithm. Pressure-based approaches were first designed for low-speed incompressible flows, while density-based approaches were primarily employed for high-speed compressible flows. Both methods, however, have lately been extended and reformulated in recent years. They are now capable of solving and operating for a much wider variety of flow circumstances than they were originally intended to. Using either method, ANSYS FLUENT will solve the governing integral equations for the conservation of mass, momentum, and (when appropriate) for energy and other scalars such as turbulence and chemical species [1]. Thus, 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 [1][3][10]. The input data carried out by the software, as shown in Figure 8 for a pressure-based solver setting and Figure 10(a) for a Pressure-Velocity Coupling (Coupled Scheme) setting.

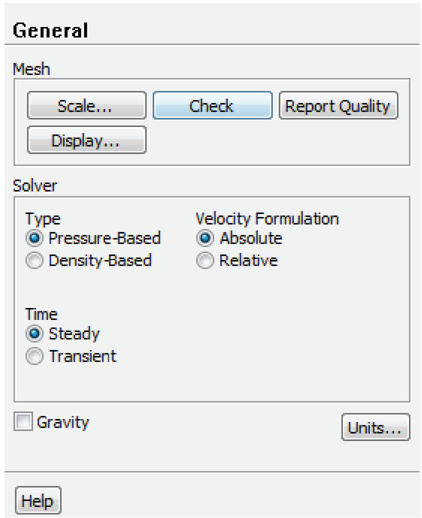


Figure 8: Pressure based Solver Setting

The physical flow phenomena indicate the presence of fluctuating velocity field. Those fluctuating phenomena mixed the transport quantities such as momentum, energy, and species concentration resulting in the transported quantities to fluctuate. To count such fluctuated quantities computationally, it is too expansive since those quantities have small scales and high frequency. To simplify the computational effort due to small scale and high frequency fluctuating flow quantities, one may use a time-averaged concept or ensemble average. These two concepts allow the governing equation of fluid motion can be simplified, but the modified governing equation produces a bunch of additional unknown variable cannot be avoided. These additional unknown variables need to be defined and known as a turbulence model [11][12].

Therefore, in view of the turbulence model available in ANSYS-Fluent software, the software provided nine main turbulent models. Some of them have more than one variant turbulence models such as k-ε models or the k-ω models, and some of them is a modification of RANS models such as Detached Eddy Simulation or DES. 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 a one-equation turbulent model, used a Boussinesq hypothesis in solving a transport equation for the kinematic eddy turbulent viscosity parameter, $\tilde{v}$. While the turbulent model called *k-ε* Standard, *k-ε* Realizable, *k-ω* Standard and *k-ω* SST represent the two-equation model. Details of various turbulent models can be obtained in [13-18].

In setting the fluid material and boundary condition to the case of flow past through the rectangular wing models, the third step of the ANSYS-Fluent setup is presented in Table 6. The difference for both flow condition is their Mach number, where for the ONERA M6 wing test case, the Mach number is 0.84, while for the RSW wing model, the Mach number is 0.802. The reference values in step four need to be defined, as shown in Figure 9.

Table 6: Type and setting value at the Boundary Condition

Boundary	Type	Condition	Viscosity (kg/ms)
Inlet Outlet Far-Side	Pressure Far-Field	Temperature = 300 K Mach Number = 0.802 Pressure = 14.7 kPa	1.846e-05
Near_Side	Symmetry	Atmospheric Pressure = 0	1.846e-05
Wing_Surface Wing_Tip	Wall	Atmospheric Pressure = 0	1.846e-05

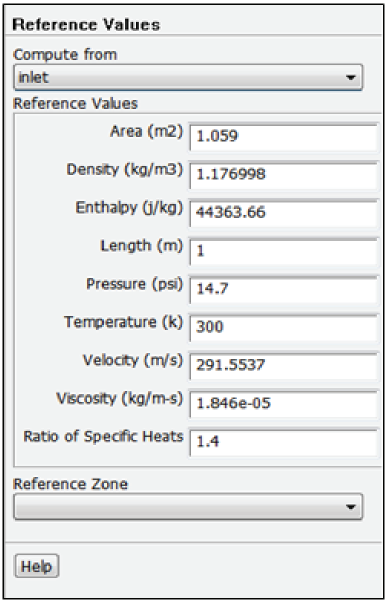


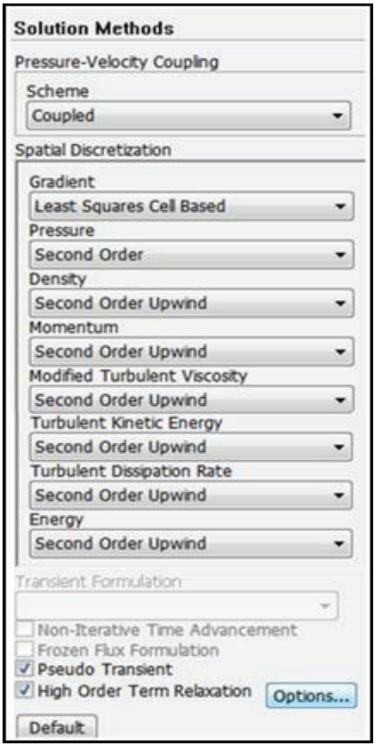
Figure 9: Reference Values Setting of the Wing Model

D. Solution Method and Calculation Task

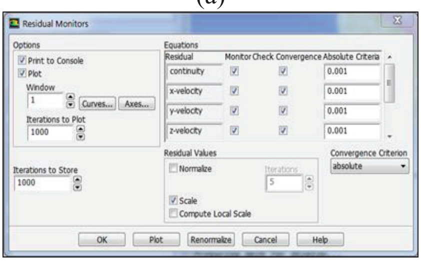
In stage four of ANSYS-Fluent software implementation, there are four steps that need to be carried out. They are namely, (1) Solution Method, (2) Monitors, (3) Solution Initialization and (4) Run Calculation.

In solution methods in which the pressure-based solver + Coupled scheme is used, a spatial discretization input needs to be defined, as shown in Figure 10(a). It is necessary to be noted here that the residual that appears in solution calculation is the error of magnitudes for equations as iterations progress. According to theory, the residual should reach zero values as the solution converges; however, in the actual calculation, a non-zero value is impossible to achieve. Therefore, the values of residuals will decay to some small value ("round-off") and then stop changing ("level out"). In this present work, the Scaled Residual monitor for both wing models 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} [1] as shown in the residual monitor in Figure 10(b).

In the next step for both wing models, a Solution Initialization is needed for the solver to start with its first iteration process. A Hybrid Initialization is used as a first initial value for the flow variables in every grid cell by using a Laplace Equation for solving the flow variable for velocity and pressure field. The setting of a boundary condition on the wings surface is used as a case of the External-Aero problems.



(a)



(b)

Figure 10: (a) Solution Methods for Pressure-Velocity Coupling and Spatial Discretization settings; (b) Residual Monitors

E. Analysis of Result

Once all the first four main phases are completed, the real calculation process can be carried out. The result and discussion for different turbulent models are presented in the following subchapter.

IV. Result and Discussion

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 work, an ONERA M6 wing model, which represents the swept-wing planform and rectangular supercritical wing, has been evaluated experimentally for the purpose of comparing it to a turbulence model available in the ANYSY-Fluent software.

For a first model, two experimental flow condition of the swept wing is retrieved from AGARD, namely at the angle of attack, $\alpha = 3.06^o$ and $\alpha = 6.06^o$ in the flow with Mach number $M_\infty = 0.84$ and the Reynolds number $Re = 11.76 \times 10^6$. The best turbulence model is chosen from the analysis of swept-wing planform for later use in the next simulation.

The second model, which known as rectangular supercritical wing planform, the geometry and experimental data, is provided by RTO-TR-026 [4][5]. The model is tested at a flow condition, namely, at the angle of attack, α = 2.00º in the flow with Mach number M_∞ = 0.802 and the Reynolds number Re = 0.401×10^7.

A. Test Case of Moderate Swept Wing Planform (ONERA M6 Wing)

Two wind tunnel test over ONERA WING M6 is known as test case number 2308 and 2565 [2], are used for a comparison purpose. These two experiments are conducted at the Reynolds number, Re, Mach number, M_∞ and the angle of attack, α, as shown in Table 7.

The comparison result between ANSYS Fluent software with different turbulent models and the experimental in term of lift coefficients C_L, their differences and the required iteration number for the ANSYS-Fluent software converge for the test case number 2308 and 2565 are shown in Table 8.

Table 7: ONERA M6 Wing Planform 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

Table 8: Absolute Relative Errors (%) 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 = 3.07º			Number of Iteration Step
	C_L A-F	C_L Exp	C_L of ABS (%)	
Spalart-Allmaras	0.2483	0.2583	3.887	60
k-ε Standard	0.2491	0.2583	3.59	78
k-ε Realizable	0.2485	0.2583	3.806	212
k-ω Standard	0.2485	0.2583	3.795	60
k-ω SST	0.2457	0.2583	4.878	60
Turbulence Model	**Test Case 2565 Angle of Attack = 6.07º**			**Number of Iteration Step**
	C_L A-F	C_L Exp	C_L of ABS (%)	
Spalart-Allmaras	0.5012	0.4908	2.112	62
K-ε Standard	0.5046	0.4908	2.806	74
k-ε Realizable	0.4784	0.4908	2.533	357
k-ω Standard	0.4824	0.4908	1.712	113
k-ω SST	0.502	0.4908	2.289	62

where:
A-F represents ANSYS-Fluent Simulation; ABS represent absolute relative error; Exp represents experiment results.

Considering the comparison result, as shown in Table 8, one can conclude that ANSYS Fluent software with all five turbulence models is in good agreement with the experimental results. These five turbulence models can be used for analyzing flow past through swept wing. However, if one looks at the required iteration number, it can be said that the Spalart-Allmaras and *k-ω* SST turbulence model represents the best choice for the turbulence model as the number of iteration steps is among the smallest. These two-turbulence model is later used for comparison purpose of a test case of the rectangular supercritical wing model.

B. Test Case of Rectangular Supercritical Wing Planform

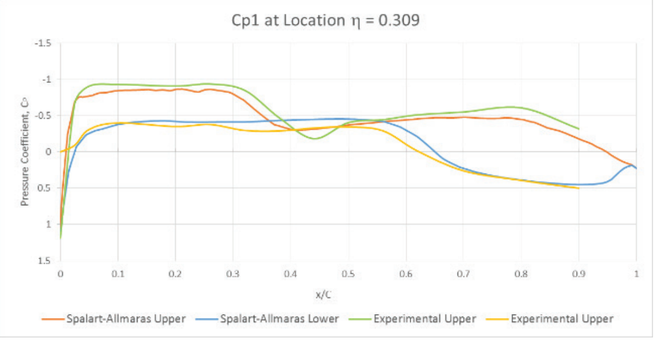
The geometry and experimental data for a straight wing are provided by RTO-TR-026. The test case represents the wind tunnel test over Rectangular Supercritical Wing or RSW, as shown in Table 9 [4][5].

The comparison results in term of pressure coefficient distribution, C_p at different spanwise locations between ANSYS-Fluent software with Spalart-Allmaras and experimental result, as shown in Figure 11, while for the *k-ω* SST turbulence model and experiment, as shown in Figure 12.

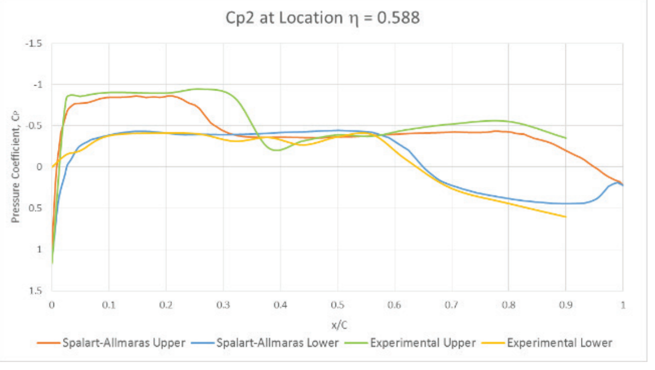
Considering the comparison result in Figure 11, the Spalart-Allmaras turbulence model generates the result at those four spanwise locations that are relatively close each to other. The same tendency can be found as well for the *k-ω* SST turbulence model with the comparison result, as shown in Figure 12. Strictly speaking, the Spalart-Allmaras and the *k-ω* SST turbulence model represents the turbulence model, which make the ANSYS Fluent software are able to produce the result close to the experimental result for wing models.

Table 9: Rectangular Wing Planform Experimental Flow Condition

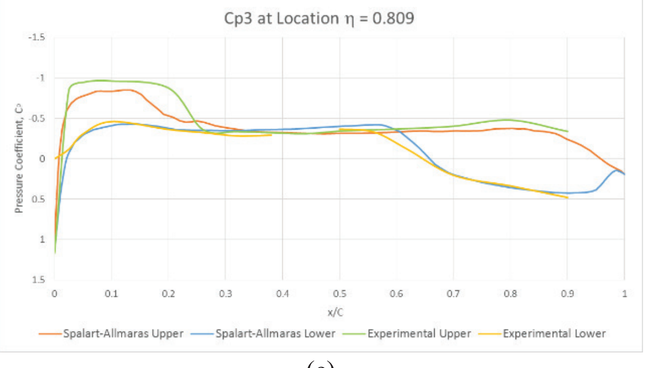
Test Number	Reynolds Number, (Re)	Mach Number, (M_{∞})	Angle of Attack, α (degree)
RSW	4.0×10^6	0.84	3.06^0



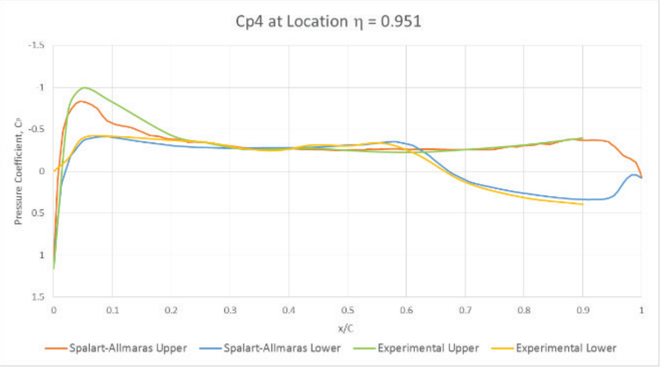
(a)



(b)

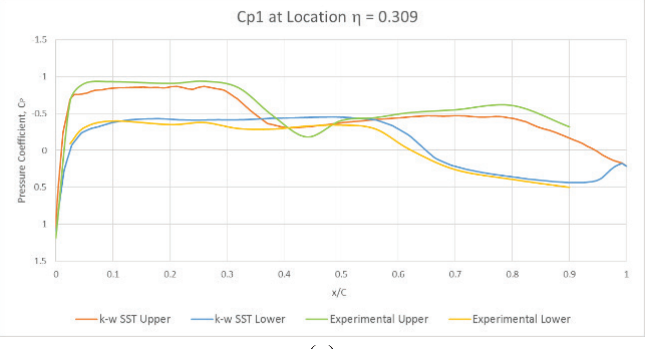


(c)

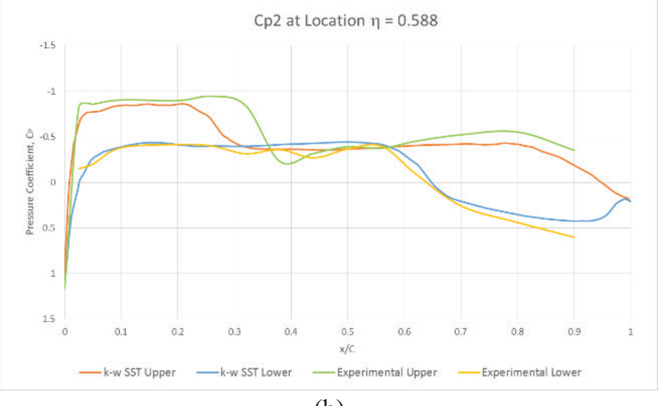


(d)

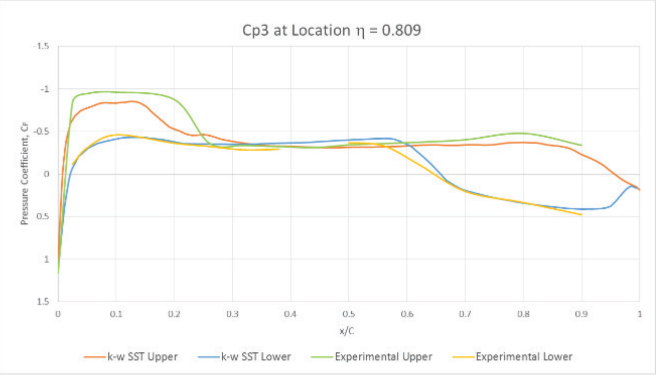
Figure 11: Comparison result of straight-wing pressure coefficient distribution, C_p along the chord at four spanwise stations between experimental result and ANSYS-Fluent simulation (Spalart-Allmaras turbulence model)



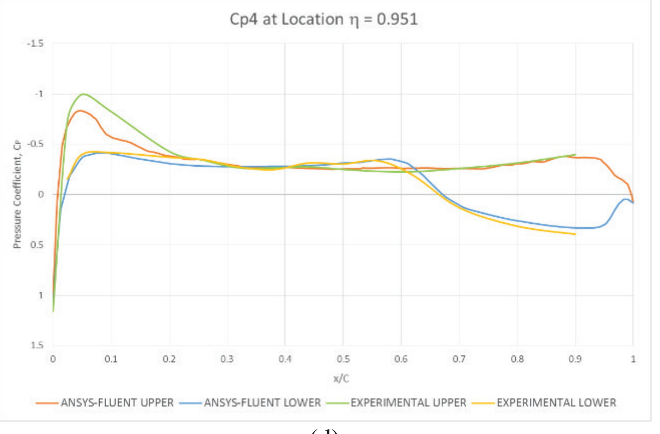
(a)



(b)



(c)



(d)

Figure 12: Comparison result of straight-wing pressure coefficient distribution, C_p along the chord at four spanwise stations between experimental result and ANSYS-Fluent simulation (k-ω SST turbulence model)

Table 10: Absolute Relative Errors (%) of Lift Coefficient, C_L for Spalart-Allmaras and k-ω SST Turbulence Model

Turbulence Model	Test Case 2308 - Angle of Attack = 2.00º		
	C_L A-F	C_L Exp	C_L of ABS (%)
Spalart-Allmaras	0.327	0.2866	14.5
k-ω SST	0.32	0.2866	11.5

However, in view in term of lift coefficient C_L which this value is obtained from the integration of pressure coefficient C_p at those four-sections gives their value and their difference with the experimental result as shown in Table 10. The Spalart-Allmaras turbulence model produces different results, which is 14.05% compared to the experimental result. Meanwhile, for the *k-ω* SST turbulence model, the comparison result for experimental and ANSYS-Fluent simulation is 11.5%. Such difference may present due to the number of spanwise sections are too small. However, this result is acceptable as the CFD method should provide a discrepancy of less than 15% and a similarity of the flow characteristics to the experiment [19].

V. Conclusion

Considering the comparison result with the experimental result as presented in the result and discussion, it can be concluded that the Spalart-Allmaras and the k-ω SST turbulence model, when combined with a pressure-based solver, can be used for a compressible and low Mach number aerodynamic analysis. These two turbulence models are able to produce the result less than 15% to the experimental result.

Further studies on the highly swept wing should be considered for future works alongside the case of the various angle of attack. In particular, a grid-independent study should be conducted in the future in understanding the effect of the grid meshing on the turbulence models. The purpose of these future studies is to deeply understand the flow pattern and the capability of the turbulence model in ANSYS software in solving the flow problem at any kind of wing models and flow conditions.

VI. References

[1] A. Fluent, “ANSYS Fluent Theory Guide Version 16.0,” 2015.

[2] V. SCHMITT 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*, no. 138, 1979.

[3] M. Z. M. Shah, B. Basuno, and A. Abdullah, “Comparative Study on Several Type of Turbulence

Model Available in ANSY-Fluent Software for ONERA M6 Wing Aerodynamic Analysis," *J. Adv. Mech. Eng. Appl*., vol. 1, no. 1, pp. 9–19, 2020.

[4] R. M. Bennet, "Test Cases For a Rectangular Supercritical Wing Undergoing Pitching Oscillations," *Verification and Validation Data for Computational Unsteady Aerodynamics*, 2000.

[5] R. H. Ricketts, M. C. Sandford, D. A. Seidel, and J. J. Watson, "Transonic Pressure Distributions on a Rectangular Supercritical Wing Oscillating in Pitch.," *J. Aircr*., vol. 21, no. 8, pp. 576–582, 1984.

[6] H. K. Versteeg and W. Malalasekera, "Introduction to Computational Fluid Dynamics," vol. M. 2012.

[7] P. K. Kanti and C. Shyam, "A Review Paper on Basics of CFD and Its Applications," *Int. J. Eng. Trends Technol*., no. NCAMES, pp. 257–264, 2016.

[8] D. Anderson, J. C. Tannehill, and R. H. Pletcher, "Computational fluid mechanics and heat transfer," *Third edition*, 2016.

[9] H. H. Fernholz, "Boundary Layer Theory: (8th (English) revised and enlarged edition) by H. Schlichting, K. Gersten (Springer-Verlag Berlin, Heidelberg, New York, 2000, pp. 799.

[10] F. J. Kelecy, "Coupling Momentum and Continuity Increases CFD Robustness," *ANSYS Advantage*, Volume II, Issue 2, 2008.

[11] T. Cebeci and P. Bradshaw, "Solutions Manual and Computer Programs for Physical and Computational Aspects of Convective Heat Transfer," 2013.

[12] R. Rubinstein, "Tuncer Cebeci: Analysis of Turbulent Flows," *Theoretical and Computational Fluid Dynamics*, vol. 19, no. 4. pp. 301–302, 2005.

[13] P. R. Spalart, S. R. Allmaras, and J. Reno, "One-Equatlon Turbulence Model for Aerodynamic Flows," *30th Aerosp. Sci. Meet. Exhib*., pp. 23, 1992.

[14] D. B. S. B. E. Launder, "The Numerical Computation of Turbulent Flows," *Comput. Methods Appl. Mech. Eng*., vol. 3, no. 2, pp. 269–289, 1974.

[15] Z. Y. and J Z. T. H. Shih, W. W. Liou, A. Shabbir, "A New k-e Eddy Viscosity Model for High Reynolds Number Turbulent Flows-Model Development and Validation," *NASA Tech. Memo*., no. 106721, 1994.

[16] D. C. Wilcox, Turbulence Modelling for CFD, 3rd Edition, 2006.

[17] F. R. Menter, "Two-equation eddy-viscosity turbulence models for engineering applications," *AIAA J*., vol. 32, no. 8, pp. 1598-1605, 2008.

[18] F. R. Menter, "Review of the shear-stress transport turbulence model experience from an

industrial perspective," *Int. J. Comut. Fluid Dyn*., vol. 23, no. 4, pp. 305-316, 2009.

[19] S. Z. Shuja and M. A. Habib, "Fluid flow and heat transfer characteristics in axisymmetric annular diffusers," *Comput. Fluids*, vol. 25, no. 2, pp. 133-150, 1996.