Mechanical Testing and Diagnosis
ISSN 2247 – 9635, 2021 (XI), Volume 3, pp. 5-9

ANALYSIS OF VARIOUS TURBULENCE MODELS FOR NACA 23012 AIRFOIL

Constantin Cristian ANDREI[1,2] Raluca BĂLAȘA[1,2]

[1]National Institute for Aerospace Research "Elie Carafoli" – INCAS Bucharest, Romania
[2]Politehnica University of Bucharest, 313 Splaiul Independenței, Bucharest, Romania

E-mail: andrei.cristian@incas.ro

ABSTRACT

This paper compares Reynolds Averaged Navier-Stokes (RANS) turbulence models and evaluates the NACA 23012 airfoil based on these test situations. ANSYS-Fluent version 22.1 was used to compare turbulence models for high-fidelity CFD simulations. A turbulence model should be used to anticipate turbulence's effect on flow. The turbulence model is a set of constitutive equations used to close the flow-governing Navier-Stokes equations. Most engineering turbulence models are based on Boussinesq hypothesis (Spalart – Allmaras, k-ω Standard, k-ω SST, k-ε Standard, k-ε Realizable and k-ε RNG) because it gives a low-cost calculation for solving turbulence viscosity. In this work, three representative turbulence models are used: Spalart-Allmaras, k-ω SST, and k-ε Standard. The flow is investigated at various angles of attack (AoA), from -2° to 18°. These AoA correlate to $Re = 3\times10^6$ *and Mach number* $M_\infty = 0.13$*. Simulations are steady-state and incompressible. Flow velocity, pressure, and density are not time dependent. Comparing these CFD scenarios to Abbott's [2] experimental data reveals that these turbulence models offer close results with a 5% error margin for low and medium AoA.*

Keywords: ANSYS-Fluent, RANS-based turbulence model, Spalart-Allmaras, k-ω SST, k-ε Standard

1. INTRODUCTION

An airfoil with a design lift coefficient of 0.3, the point of maximum camber located at 15% of chord, and a maximum thickness that is 12% of chord length is described by the NACA 23012 airfoil, which is a member of the NACA 5-digit series of airfoils [1]. This particular airfoil was developed by NACA in the 1930s, and since then, there was no significant amount of testing and research carried out in wind tunnels. Researchers Abbott and van Doenhoff [2] glanced into a dozen different NACA wing sections, one of which was the NACA 23012. In this paper, the model geometry is created with the use of ANSYS Design Modeler, the meshing process is defined with the assistance of ANSYS Meshing, and the ANSYS Fluent software is applied to solve the CFD problem.

ANSYS is a multi-purpose software system that was developed to analyze and solve a wide variety of engineering problems faced with a range of engineering fields, such as fluid flow, structural analysis, industrial machineries, heat transfer, turbulence, and explicit dynamics.

Since the 1970s, a great number of ANSYS versions have been released, and a great number of engineering features have been added in order to handle day-to-day engineering issues. ANSYS-Fluent is the component of ANSYS that was developed specifically to address issues relating to CFD. In this work, version 22.1 of the ANSYS-Fluent software is utilized.

This software application was developed with the intention of resolving flow issues by applying the generalized Navier-Stokes equations in the form of Reynolds Averaged Navier-Stokes (RANS) equations as its governing equations of the averaged flow quantities and by applying a turbulence model in order to solve the turbulent flow parameters. Both concepts were derived from the Reynolds Averaged Navier-Stokes equations.

2. MODELING OF NACA 23012

2.1. Geometry Definition

Using the coordinate parameters that were received from the Airfoil Tools website [1], the geometry of the NACA 23012 airfoil was generated in ANSYS Design Modeler V22.1 (2017). After that, a flow domain with a C-topology was constructed. It extended ten chord lengths upstream of the NACA 23012 airfoil and fifteen chord lengths downstream of the airfoil.

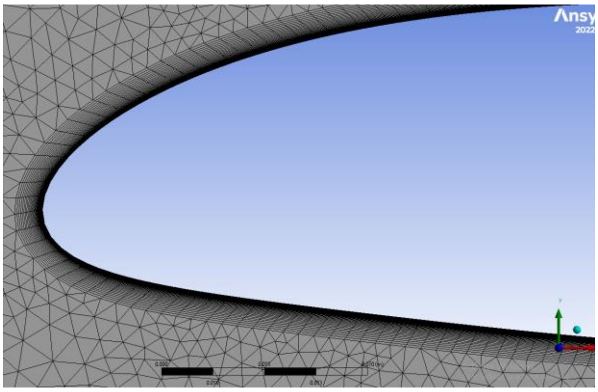

Fig. 1 – NACA 23012 leading edge mesh view, for $y^+ = 1$

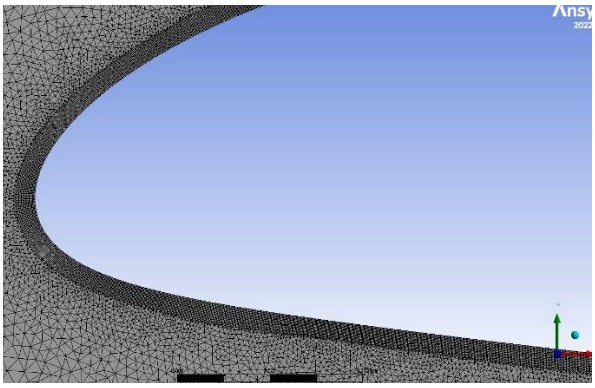

Fig. 2. NACA 23012 leading edge mesh view, for $y^+ = 100$

The completed geometry was imported into ANSYS Meshing V22.1 in order for the mesh to be generated.

2.2. Meshing

In computational fluid dynamics (CFD), the generation of the mesh plays an essential part in the calculation of the numerical solution. Using the All Triangles Method, two unstructured meshes are generated to serve as the domain definition for the mesh flow. With the exception of the inflation that is used to reproduce the boundary layer, the grids are almost identical. The initial mesh was constructed in such a way that ANSYS-Fluent, utilizing the Spalart-Allmaras and k-ω SST turbulence models, could calculate the numerical solution to the problem. In order to make it work, y^+, the value of which is a non-dimensional wall distance, was changed to 1 so that the viscous sub-layer could be included in the near body surface. This indicates that the thickness of the first layer of the boundary layer is equivalent to 1.68×10^{-5} m on the logarithmic scale, as shown in Table 1. 1.08 is what has been decided upon for the growth rate. In the second mesh, the value of y+ was changed to 100 so that the k-ε Standard turbulence model could be used by ANSYS-Fluent to find the numerical solution. The thickness of the boundary layer's first layer is calculated to be 1.68×10^{-3} m.

In conclusion, the NACA 23012 airfoil meshes each have a secondary, finer mesh that was designed to replicate the wake that was produced downstream of the airfoil. When determining the numerical solution, this finer grid is used as it produces results that are more accurate.

Both Figure 1 and Figure 2 present a view of the leading-edge mesh for the cases where $y^+=1$ and $y^+=100$, respectively.

2.3. ANSYS-Fluent Setup

Using the generalized Navier-Stokes equations as the governing analysis of fluid motion, both grids are imported into ANSYS-Fluent in order to solve the flow problem and determine the flow parameters. This is performed by using the generalized Navier-Stokes equations. The simplistic Navier-Stokes equations are time-averaged, resulting in Reynolds Averaged Navier-Stokes (RANS) equations. These equations are used to describe a flow that is turbulent [3]. These equations, which are written in a Cartesian tensor form and describe the laws of continuity, momentum, and energy respectively, are as follows:

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

$$\frac{\partial}{\partial t}(\rho u_i) + \frac{\partial}{\partial x_j}(\rho u_i u_j) = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j}\left(\tau_{ij} - \overline{\rho u_i' u_j'}\right) \tag{2}$$

$$\frac{\partial}{\partial t}(\rho E) + \frac{\partial}{\partial x_j}(\rho u_j H) = = \frac{\partial}{\partial x_j}\left(\tau_{ij}u_i + \overline{u_i'\tau_{ij}} - \overline{\rho u_j' H} + k\frac{\partial T}{\partial x_j}\right) \tag{3}$$

RANS equations (1), (2), and (3) are the mathematical form that represent the continuity (1), momentum (2), and energy (3) principles, respectively. The final term in equation (2) $-\rho\overline{u_i' u_j'}$, is an additional term that is referred to as the Reynolds Stress, and it depicts the results of turbulence [3]. The Boussinesq method is utilized in order to find solutions for RANS equations.

Table 1. Meshing Setup

	Turbulence model		
	Spalart-Allmaras	**k-ω SST**	**k-ε Standard**
y^+	1	1	100
First-layer thickness of the boundary layer (m)	1.68×10^{-5}	1.68×10^{-5}	1.68×10^{-3}
No of cells	110,000	110,000	195,000

The Reynolds Stress has the following relation, in accordance with the Boussinesq hypothesis:

$$-\rho\overline{u_i{}'u_j{}'} = \mu_t\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} - \frac{2}{3}\frac{\partial u_k}{\partial x_k}\delta_{ij}\right) - \frac{2}{3}k\delta_{ij} \quad (4)$$

The turbulent viscosity μ_t, which can be derived from equation (4) can be calculated with the use of a turbulence model. In this particular research study, the following turbulence models were utilized: Spalart-Allmaras, k-ω SST and k-ε Standard. These models, which are available in ANSYS-Fluent, are developed in accordance with the Boussinesq methodology.

The Spalart-Allmaras turbulence model is a one-equation RANS-based turbulence model that solves a transport equation for the turbulent viscosity parameter, $\tilde{\nu}$ [4] [5]. This model is an example of an equation-based turbulence model [5]. In contrast to the Spalart-Allmaras turbulence model, the k-ω SST and k-ε Standard models are RANS-based turbulence models that each consist of two equations. Both the turbulent kinetic energy and the specific dissipation are transport variables in the k-ω SST turbulence model. The turbulent kinetic energy is what determines the turbulence energy, and the specific dissipation is what reproduces the turbulence scale [6]. The turbulent kinetic energy, represented by, as well as the turbulent dissipation, represented by, are the k-ε transport variables, and they are liable for determining the turbulence scale [5] [6].

Setting the boundary conditions and reference values is required before attempting to calculate the flow parameters. Unless researchers expect that the flow is incompressible, then the boundary conditions are shown in Table 2, and the reference values for the inlet are shown in Table 3.

Table 2. Boundary Condition Setup

Boundary	Type	Condition
Inlet	Velocity – Inlet	Velocity = 44 m/s Temperature = 300 K
Outlet	Pressure – Outlet	Pressure = 101.325 kPa
Airfoil	Wall	Atmospheric Pressure = 0

Table 3. Inlet reference values

Reference Values	
Area (m^2)	1
Density (kg/m^3)	1.225
Enthalpy (J/kg)	0
Length (m)	1
Pressure (Pa)	0
Temperature (K)	300
Velocity (m/s)	44
Viscosity (kg/m·s)	1.7894×10^{-5}
Ratio of Specific Heats	1.4

In the section titled "Solution Method," the Pressure-Velocity Coupling is accomplished through the use of the Coupled scheme. This algorithm solves both the continuity equation and the momentum equation in coupled form, which eliminates the approximations that were obtained by solving the equations separately [7].

After the initialization of the solution, the calculation procedure is carried out for each turbulence model over a range of AoA that extends from -2° to 18°.

3. RESULTS & DISCUSSIONS

After the process of calculation has been completed, a comparison of the outcomes in terms of the lift coefficient for three distinct types of turbulence model is carried out. Table 4 exemplifies the results of the CFD simulations performed. As shown in Table 4, the absolute relative error for any and all of the results generated by turbulence models is no more than 5% for low and medium AoA when compared to the results obtained through experimentation. This suggests that the ANSYS-Fluent software is a reliable software application that can produce results that are comparable to those obtained through experimentation.

Table 4. Lift coefficient results

		Spalart-Allmaras		k-ω SST		k-ε Standard	
AoA	Abbott et al.	C_L ANSYS-Fluent Simulation	Absolute relative error (%)	C_L ANSYS-Fluent Simulation	Absolute relative error (%)	C_L ANSYS-Fluent Simulation	Absolute relative error (%)
-2	-0.09	-0.09102	1.13333333	-0.09035	0.38888888	-0.09039	0.433333333
0	0.13	0.12942	0.44615384	0.12765	1.80769230	0.12992	0.061538462
2	0.35	0.34982	0.05142857	0.34527	1.35142857	0.34908	0.262857143
4	0.55	0.56846	3.35636363	0.5602	1.85454545	0.56712	3.112727273
6	0.78	0.76735	1.62179487	0.76936	1.36410256	0.78053	0.067948718
8	1	0.9874	1.26	0.96784	3.216	0.98606	1.394
10	1.19	1.17888	0.93445378	1.15068	3.30420168	1.18106	0.751260504
12	1.38	1.35209	2.02246376	1.31541	4.68043478	1.35803	1.592028986
14	1.5	1.49095	0.60333333	1.43531	4.31266666	1.4556	2.96
16	1.62	1.53024	5.54074074	1.4152	12.6419753	1.44976	10.50864198
18	1.09	1.19124	9.28807339	1.23555	13.3532110	1.30875	20.06880734

Figure 3 illustrates how the polar curves of each turbulence model compare to one another. As has been observed, the lift increases along with the angle of attack up to the point where it reaches its maximum lift coefficient. When the stall effect occurs, the angle of attack that corresponds to the maximum value for the lift coefficient is 16 degrees.

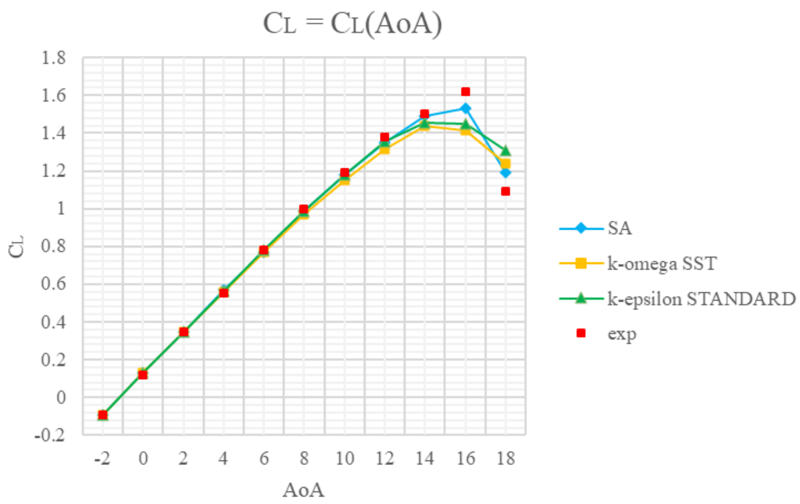

Fig. 3 – Polar curves for each turbulence model

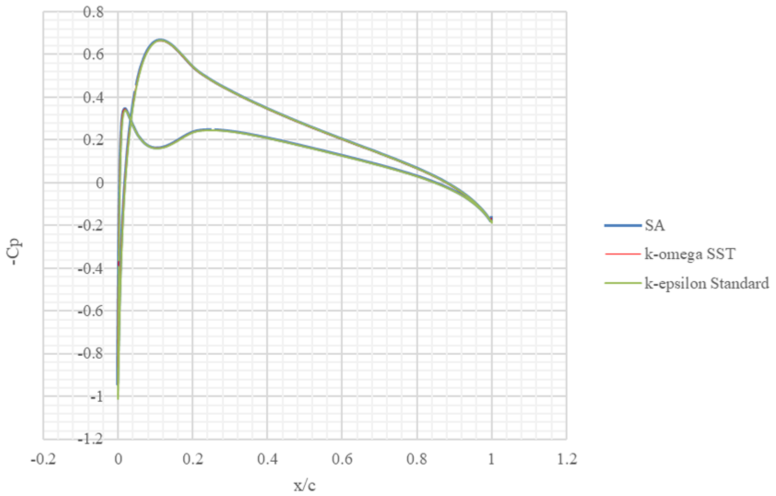

Fig. 4. Pressure coefficient distribution for AoA = 0°

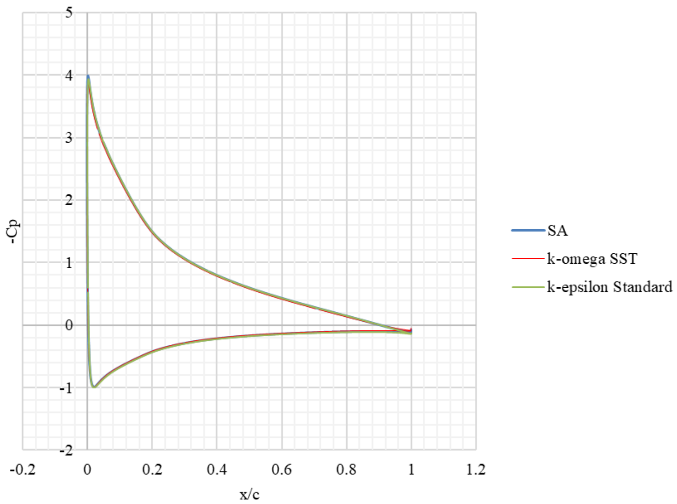

Fig. 5 – Pressure coefficient distribution for AoA=10°

The results of a comparison of the pressure coefficient distribution between ANSYS-Fluent turbulence models are shown in Figures 4 and 5 for AoA values of 0 degrees and 10 degrees, respectively. According to these figures, the pressure coefficient distribution for each turbulence model is nearly identical, which is the factor why the models produce comparable outcomes. As a result, there is only potential for consideration of a single turbulence model when researching the pressure coefficient distribution.

Even though the pressure coefficient for the upper surface of the airfoil is lower than the pressure coefficient for the lower surface of the airfoil, the lift force, which can be interpreted as the pressure difference between the airfoil's upper and lower surface, is exerted in the direction of upward orientation. According to the findings noticed, the pressure coefficient difference between the upper and lower surface is significantly greater when the AoA value is higher. As a direct consequence of this, the pressure differential between the upper and lower surface continues to widen, which in turn leads to an increase in lift.

4. CONCLUSIONS

A numerical analysis of a NACA 23012 airfoil with three different turbulent models was performed with the help of the software ANSYS-Fluent. In the current study, the following turbulence models were utilized: Spalart-Allmaras, k-ω SST, and k-ε Standard. Both ANSYS Design Modeler and ANSYS Meshing were used to model the flow domain. ANSYS Meshing was used to perform the grid. In order to produce more precise results, a mesh refinement was performed in close proximity to the airfoil.

The results for the lift coefficient obtained by ANSYS-Fluent were compared with those obtained by Abbott et al. for NACA 23012. This was done for each angle of attack [2]. Because the relative absolute error between the numerical solution and the experimental solution is less than 5% for low and medium AoA, as demonstrated in Table 4, ANSYS-Fluent is an appropriate tool for use in CFD problems.

In addition, a comparison was carried out regarding the pressure coefficient distribution for the two different cases (AoA = 0° and AoA = 10°). All of the turbulence models, when applied to each situation, produce satisfactory outcomes with no discernible variations.

REFERENCES:

1. *** Airfoil Tools, Online http://airfoiltools.com/. (accessed 23.10.2022)
2. Abbott I. H., von Doenhoff A. E. (1949). Theory of wing sections. New York : Dover Publications, Inc.
3. Dănăilă S., Berbente C. (2003) Metode numerice în dinamica fluidelor (in Romanian). Bucharest, Ed. Academiei Române.
4. Spalart P., Allmaras S. (1992). A one-equation turbulence model for aerodynamic flows. s.l.: American Institute of Aeronautics and Astronautics.
5. Dănăilă S., Stoia-Djeska M. (2017) Introducere în modelarea turbulenței. București, Ed. Politehnica Press.
6. Wilcox, D. C. (1998) Turbulence Modeling for CFD. s.l.: Anaheim: DCW Industries.
7. *** ANSYS Fluent version 22.1. s.l.: ANSYS Fluent Theory Guide, 2022.