

NUMERICAL ANALYSIS OF LIFT AND PRESSURE COEFFICIENTS OF AN AIRFOIL: ANSYS VS SOLIDWORKS

NUMERIČNA ANALIZA KOEFICIENTA VZGONA IN TLAKA PROFILA KRILA: ANSYS IN SOLIDWORKS

Igor Spaseski[✉], Marko Pezdevšek, Igor Ščuri, Matej Fike, Gorazd Hren

Keywords: numerical analysis, airfoil, CFD, Ansys CFX, SolidWorks

Abstract

In this article, curves for the lift and pressure characteristics of an NACA 6508 airfoil at various angles of attack were acquired with numerical simulation software packages: Ansys CFX and SolidWorks Flow Simulation. The analyses were performed using three different mesh resolutions, with similar number of elements, for different angles of attack. Furthermore, all numerically obtained results were compared to the experimental measurements from a reliable source.

Povzetek

V članku so predstavljeni rezultati numerične analize tlačnih koeficientov v okolici profila in koeficienta vzgona pri različnih napadnih kotih za osamljeno krilo s profilom NACA 6508. Za izvedbo numeričnih simulacij sta bila uporabljena programska paketa Ansys CFX ter SolidWorks Flow Simulation. Analiza je bila izvedena s tremi strukturiranimi mrežami različne gostote elementov za več različnih napadnih kotov. Rezultate vseh mrež smo primerjali z eksperimentalno izmerjenimi vrednostmi iz primarne literature.

[✉] Corresponding author: Igor Spaseski, Tel.: +386 31 828 490, Mailing address: Florjanska ulica 125, 8290 Sevnica, Slovenia, E-mail address: igor.spaseski@gmail.com

1 INTRODUCTION

The evolution of computational fluid dynamics (CFD) is driven by the need for faster and more accurate methods for the calculations of flow fields around configurations of technical interest. For a decade, CFD has been the method of choice in the design of many types of industries and processes in which fluid or gas flows play a significant role. In CFD, there are several commercial packages available for simulating flow in or around objects. The computer simulations show features and details that are difficult, expensive, or impossible to measure or visualize experimentally.

The first step in simulating a problem involves the creation of the geometry (model) and physical model. The majority of the time spent on a CFD project is usually devoted to generating a mesh for the domain geometry that allows a balance between the desired accuracy and solution time. After the creation of the mesh, a solver is able to solve the governing mathematical equations of the problem. Afterwards, the results are validated with experimental measurements, [1].

Many researchers have investigated flow conditions around the isolated profile, experimentally and numerically. In [2] a comparison of flow conditions between an open and closed trailing edge was analysed with Ansys CFX. The flow field around a profile at higher angles of attack was found to be time dependent. Detailed investigation of the flow phenomena requires transient simulations. An investigation of flow conditions around a profile with steady state and transient simulation using Ansys CFX was performed in [3]. The goal of our study is not to numerically investigate the flow field around a profile in detail, but to compare the results of the predicted global parameters such as the lift and pressure coefficients for the two-selected software with experimental results.

2 GOAL DEFINITIONS

In order to compare different numerical analysis software, we decided to perform numerical analyses and compare the results to experimental data. The experimental data were obtained from [4] for the lift and pressure characteristics of an airfoil NACA 6508. The comparison was performed at different angles of attack (AOA) from 0° to 20° with 2° step.

Steady-state simulations were made with Ansys CFX 15.0 and SolidWorks Flow Simulation (SWFS) software packages in order to obtain lift and pressure coefficient curves. We have to emphasize the difference of software purpose and limitations. Ansys CFX is well-known commercial standalone numerical software for numerical analyses, and SolidWorks Flow Simulation is part of CAD (Computer-aided design) package.

We attempt to create corresponding meshes and boundary conditions, and to compare the results of analyses, definitions of meshes and user-friendliness of software.

3 NUMERICAL MODEL

To obtain reliable results, we created three meshes with different but comparable densities in both numerical packages. SolidWorks Flow Simulation provides only a $k-\varepsilon$ turbulence model, while Ansys CFX offers more models, so we performed analyses with a $k-\varepsilon$ turbulence model and the shear stress transport (SST) turbulence model.

3.1 Geometry and meshing

2D geometry of NACA 6508 was modelled from [5] in SolidWorks (SW) software and with ICEM CFD 15.0. Furthermore, structural numerical meshes were also designed with this software.

The mesh creation has two primary functions. Firstly, to import characteristic points to model the airfoil and defines the size of the domain; secondly, the discretisation and mesh generation of the computational domain.

Both of the programs used do not allow calculations in 2D; therefore, a 2D numerical mesh was created and extruded into the third dimension for one element of thickness.

The square computational domain around the airfoil was discretised by a structured mesh. The size of the domain was seven lengths of an airfoil chord length ahead of the airfoil and 15 lengths behind the airfoil.

Several simulations of pressure distribution on the airfoil surface were performed to investigate the effect of mesh resolution on the results. Generally, a numerical solution is more accurate with an increased number of elements and nodes with higher densities of elements in areas of interest; consequently, the required computer memory and computational time increases. Therefore, three meshes with different numbers of elements were designed and used for performing simulations at various angles of attack.

In Figures 1 and 2, the parts of structural mesh created in ICEM CFD and SolidWorks are presented, respectively. The resolution of the mesh is greater in regions where greater computational accuracy is needed, such as the region close to the airfoil.

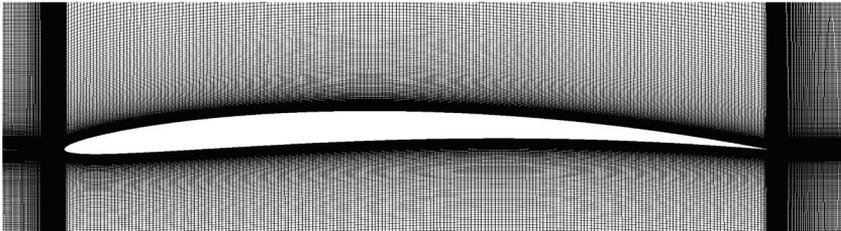


Figure 1: Numerical mesh designed with ICEM CFD software

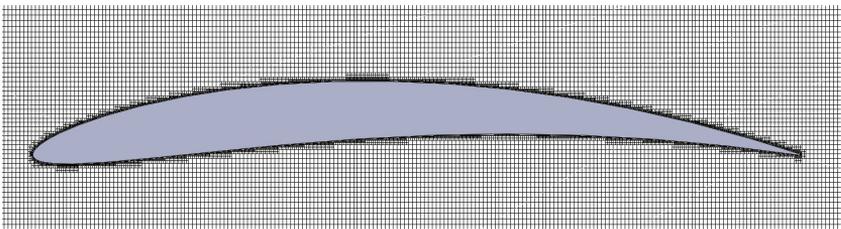


Figure 2: Numerical mesh designed with SolidWorks software

In Figure 3, the close-ups of the mesh are shown, which clearly represent a different type of mesh creation. Typical structured mesh from ICEM CFD (a) and the use of a Cartesian-based mesh generation in SolidWorks (b) [6] are presented.

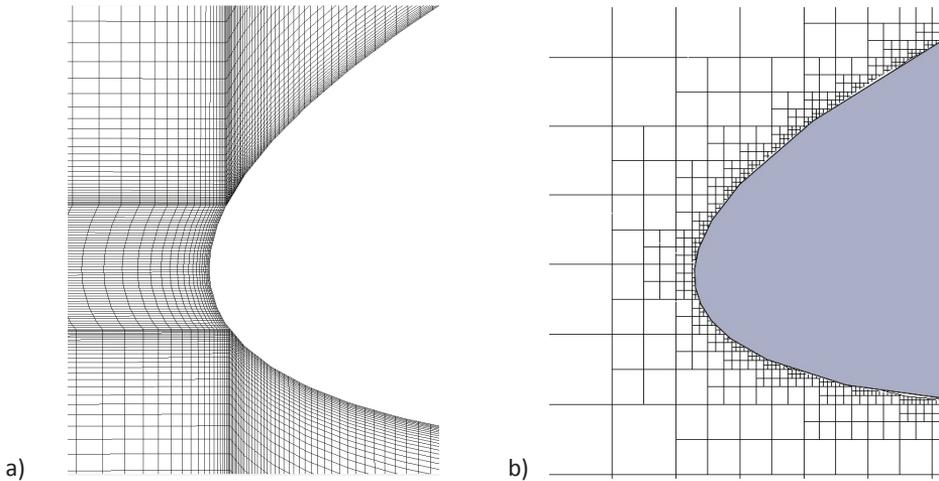


Figure 3: Numerical mesh designed with a) ICEM CFD and b) SolidWorks software in close-up view

To investigate the height of the first cell adjacent to the surface of the airfoil, a dimensionless wall distance (y^+) simulation was performed. Precise computed results were possible only if the resolution of the mesh near the wall of the airfoil satisfied the condition $y^+ < 1$. As seen in Figure 4, the value of y^+ was greater than one only at the nose and the trailing edge of the airfoil.

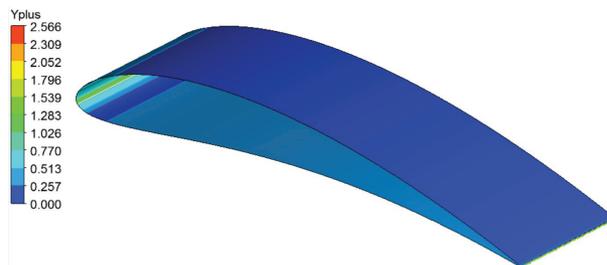


Figure 4: y^+ near airfoil wall for the medium mesh at AOA 10°

The mesh data of all three meshes are presented in Table 1.

Table 1: Mesh data

Mesh resolution	NACA 6508	
	Ansys CFX Number of elements	SolidWorks Number of elements
Coarse	77,940	84,021
Medium	307,980	344,512
Fine	517,044	490,078

3.2 Boundary conditions and convergence criteria

The numerical meshes described above were the same for all angles of attack. Therefore, the airfoil was stationary in the domain, and all AOA were defined by changing the inlet velocity vectors. The reference pressure in the domain was set to 100 kPa, and the temperature was 293 K. The following boundary conditions are defined as inlet, opening, wall and symmetry. The left and bottom sides of the domain were defined as inlets (in Ansys CFX). The upper and right side of the domain were defined as openings with a pressure of 100 kPa. For the front and backside, the symmetry boundary condition was used. The airfoil edge was defined as a no-slip wall.

The absolute inlet speed was calculated with equation:

$$v = \frac{Re \cdot \nu}{c}, \quad (3.1)$$

where:

Re - Reynolds number ($Re=100,000$);

ν - kinematic viscosity ($\nu=15.1 \cdot 10^{-6} \text{ m}^2/\text{s}$);

c - airfoil chord length.

The calculated speed at inlet was 18.88 m/s, [4]. Components of the velocity vectors v_x and v_y for the selected AOA (φ) were calculated with equations (3.2):

$$\begin{aligned} v_x &= v \cdot \cos \varphi, \\ v_y &= v \cdot \sin \varphi. \end{aligned} \quad (3.2)$$

The air density was set to 1.185 kg/m^3 , the dynamic viscosity to $1.79 \cdot 10^{-5} \text{ kg/ms}$, turbulence intensity to 1.5 % and turbulence length scale to 0.01 m, [4]. To satisfy the convergence criteria, all the RMS leftovers from solving equations must be under $1 \cdot 10^{-5}$. We set the number of maximum iterations to 500 and automatic timescale control. For the SST turbulence model at higher AOA, we used a larger physical timescale to ensure convergence. Both software packages have an automatic system for stopping the analysis when it reaches defined convergence criteria.

4 RESULTS

The computed results of the pressure distribution on the airfoil surface at different AOA were compared to existing experimental data [4], but there was no data of the lift coefficient, so the lift coefficient results between the two software packages were compared only. Based on the results seen in Figure 7 and the computational times seen in Table 2, we selected the coarse mesh for Ansys *k-ε* and medium mesh for both SolidWorks and Ansys SST.

4.1 Pressure distribution on airfoil surface

Pressure is usually presented in the form of a pressure coefficient C_p and is calculated with the equation:

$$C_p = \frac{p - p_0}{\frac{1}{2} \cdot \rho \cdot v^2}, \quad (4.1)$$

where:

p - simulated (measured) pressure;

p_0 - reference pressure (100 kPa);

ρ - air density;

v - airflow speed.

Figures 5 and 6 show the comparison of numerical analysis results and experimental data.

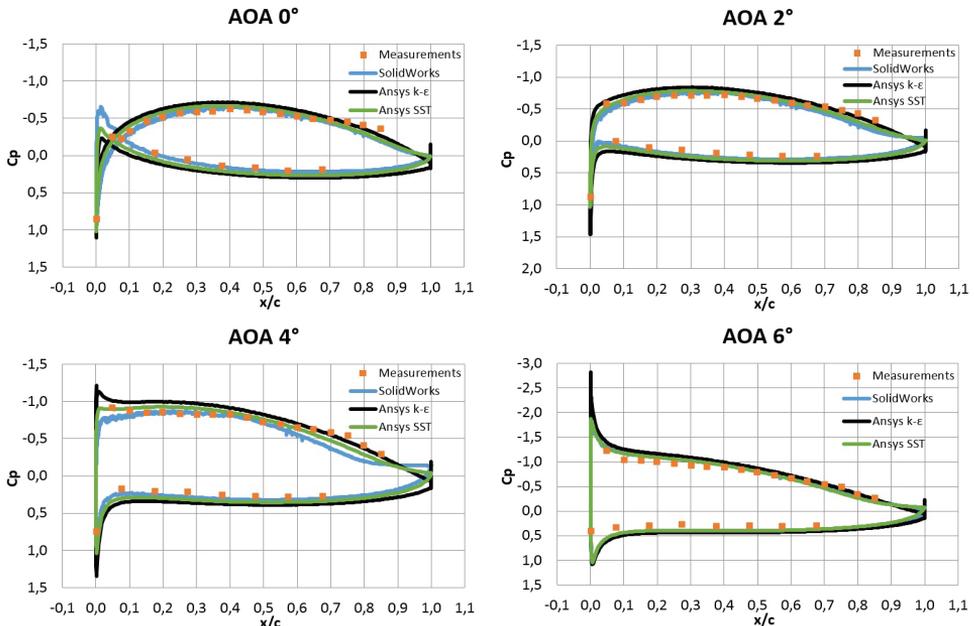


Figure 5: Comparison between experimental and CFD results of C_p at AOA ranging from 0° to 6°

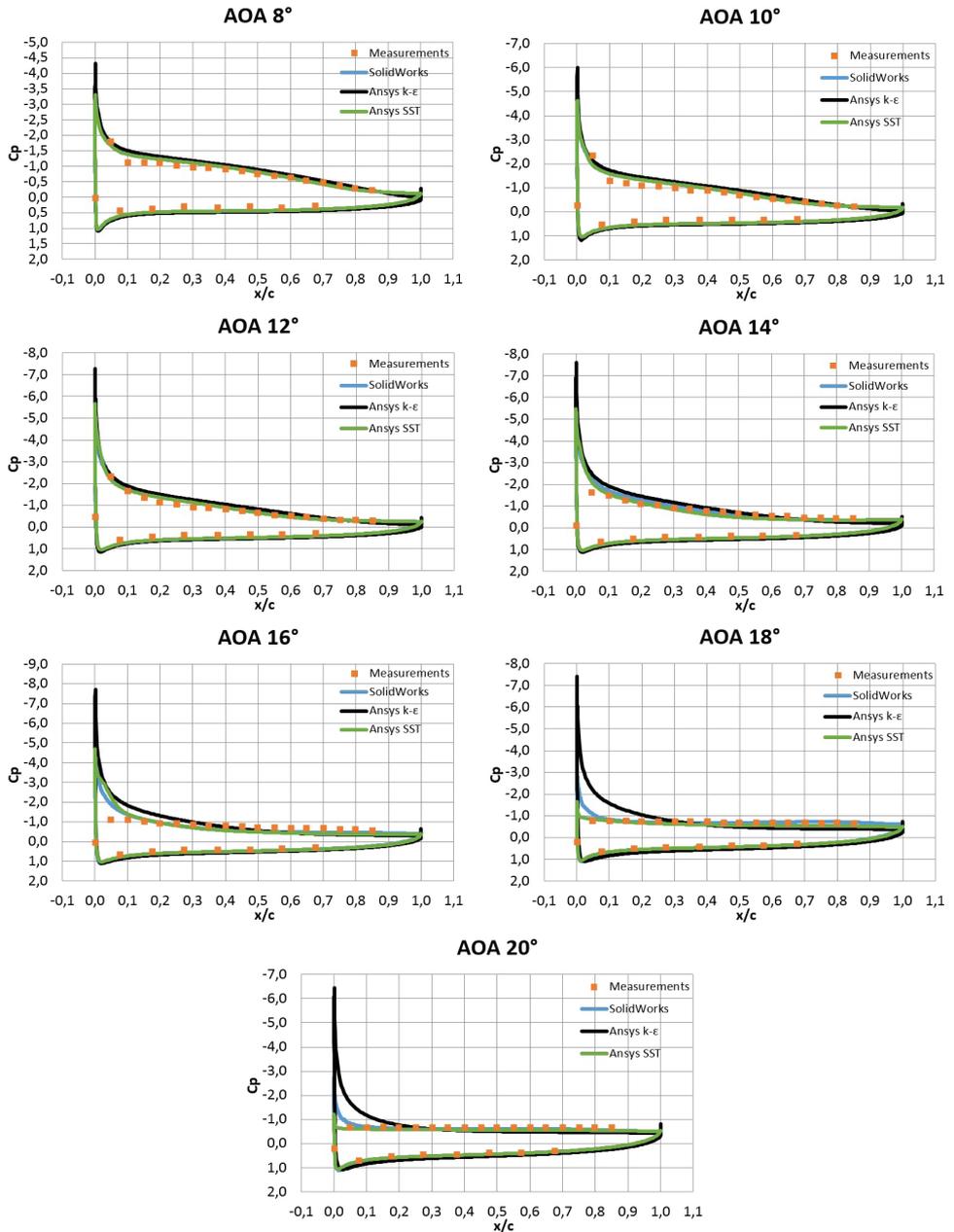


Figure 6: Comparison between experimental and CFD results of C_p at AOA ranging from 8° to 20°

4.2 Lift coefficient

From numerical simulations, we acquired forces in the horizontal and vertical directions from which we calculated the lift force. We calculate the lift coefficient C_l with the following equation:

$$C_l = \frac{F_l}{\frac{1}{2} \cdot c \cdot \rho \cdot v^2}, \quad (4.2)$$

where:

F_l - lift force;

c - airfoil chord length;

ρ - air density;

v - air speed.

Figure 7 shows the comparison of numerical analysis for lift coefficient at different AOA with various mesh resolutions.

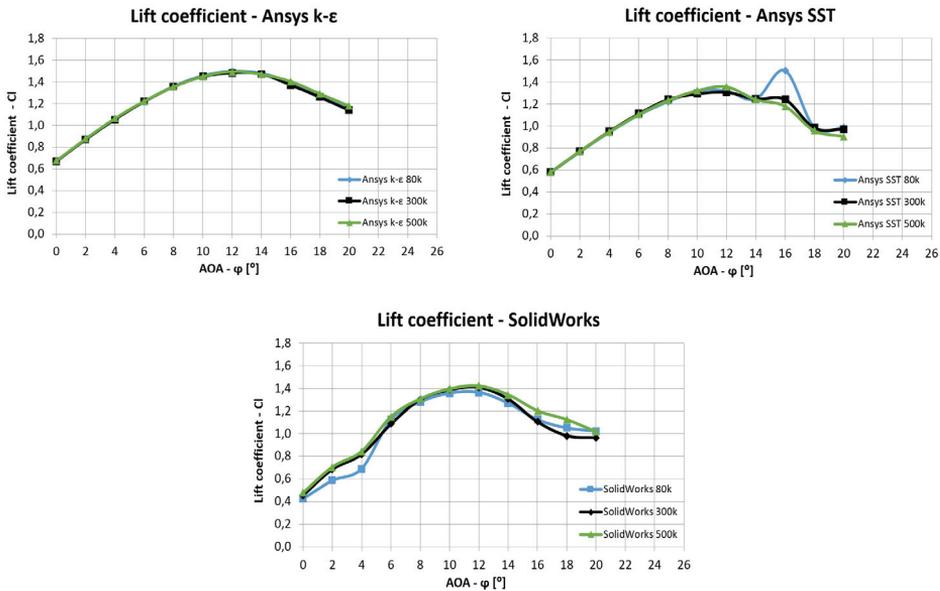


Figure 7: A comparison between numerically obtained values from both software packages of lift coefficients at various AOA for different mesh resolutions

Figure 8 shows the comparison numerically obtained values from both software packages of lift coefficients at various AOA for selected mesh resolution.

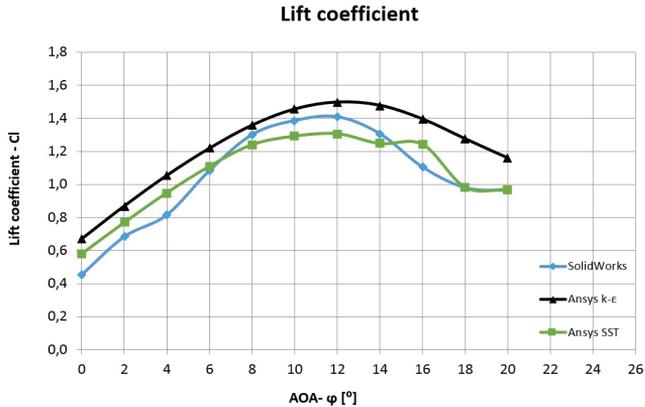


Figure 8: A comparison between numerically obtained values from both software packages of lift coefficients at various AOA for selected mesh resolution

4.3 Computation time

Considering computation times, we present computation times for AOA of 10 ° as a reference. As can be seen in Table 2, computational times in Ansys CFX software using $k-\epsilon$ turbulence model are longer in comparison to the SST model. Furthermore, the SolidWorks software took about twice as much time to conclude the simulation on a coarse mesh. With increased density of the mesh, the computation time is much longer.

Table 2: Computational times for computing pressure distribution on airfoil surface at AOA 10 °

Mesh resolution	Computational time [hh]:[mm]:[ss]		
	Ansys CFX SST model	Ansys CFX $k-\epsilon$ model	SolidWorks
Coarse	0:02:07	0:02:29	0:05:52
Medium	0:07:06	0:10:30	0:45:46
Fine	0:11:13	0:16:01	1:33:30

5 CONCLUSIONS

Simulations of pressure distribution on the airfoil surfaces and lift coefficients for various angles of attack were made in order to be able to compare the results from the different software packages and then validate them with existing experimental data from a reliable source.

The difference in accuracy of the results compared to experimental data become more apparent at higher AOA, because the flow on the upper surface of the airfoil begins to separate, and a condition known as stall develops. Different turbulence models predict this effect more or less accurately. Figures 5 and 6 show that the pressure coefficient values computed with Ansys CFX using the $k-\epsilon$ turbulence model deviates the most from the experimental data. By increasing the angle of attack, this deviation becomes even larger. From the results, we also see that the SST model most accurately describes the pressure coefficient curve. Greater deviation occurred at the angle of 16° , which is the result of the solution not converging; we see this problem with all three meshes. There were no converging problems in the SolidWorks or the $k-\epsilon$ turbulence model. Results obtained from SolidWorks are quite comparable to experimental data; SolidWorks generally does better than $k-\epsilon$ but worse than SST in Ansys.

Figure 7 shows how different mesh resolutions affect the lift coefficient curve. With the $k-\epsilon$ turbulence model, the mesh resolution does not affect results. For the SST model, all the designed meshes were sufficient to the angle of 10° ; at higher angles, the deviation between meshes was larger. The largest deviation occurred at the angle of 16° , which is the angle at which the solution did not converge. We can also see that at this angle the coarse mesh oscillates the most, and its value was about 30 % higher than the fine mesh, while the medium mesh deviated by about 5 %. The medium mesh designed with SolidWorks is sufficient between angles 0° and 14° . At higher AOA, it deviates more and achieves maximum deviation of 15 % at AOA 18° . The coarse mesh in SolidWorks is insufficient at AOA lower than 6° and higher than 14° .

Computational times in Ansys CFX software using $k-\epsilon$ turbulence model were slightly longer than those using SST model. Furthermore, the SolidWorks software took about twice as much time for performing the simulation on a coarse mesh, as the Ansys SST and $k-\epsilon$ models did. As the number of elements increases and their size decreases, the computational time in SolidWorks is as much as five times longer than in Ansys CFX. Based on the results seen on Figure 7 and the computational times seen in Table 2, we selected the coarse mesh for Ansys $k-\epsilon$ and the medium mesh for both SolidWorks and Ansys SST models. The lift coefficient curves of the selected meshes are presented in Figure 8, where we can see that the values of $k-\epsilon$ model are greater at all AOA. Local maximum of the computed curves was the same at AOA 12° .

Based on the computed results and simulation time, the Ansys SST model has proved to be the most suitable, but if we take into the account the time and effort to make a structured mesh in ICEM CFD, SolidWorks would be the better pick for the example used in this paper.

References

- [1] **D. C. Eleni, T. I. Athanasios, M. P. Dionissios:** *Evaluation of the turbulence models for the simulation of the flow over a National Advisory Committee for Aeronautics (NACA) 0012 airfoil*, University of Patras - Greece, Department of Mechanical Engineering and Aeronautics, Fluid Mechanics Laboratory (FML), 2012,
Available at: http://academicjournals.org/article/article1379753908_Eleni_et_al.pdf, Accessed on October 6, 2014
- [2] **J. Bitenc, B. Širok, I. Biluš:** *Numerical analysis of flow over a wind turbine airfoil*, Journal of Energy Technology, Volume 6, Issue 4, 2013, pp. 31–46.
- [3] **M. Fike:** *The unsteady static-stall aerodynamic characteristic of an S809 airfoil at low Reynolds numbers*, Journal of Energy Technology, Volume 6, Issue 1, 2013, pp. 33–50.
- [4] **M. Fike:** *Experimental and numerical analysis of fluid flow in an axial fan*, Doctoral thesis, University of Maribor, Faculty of Mechanical Engineering, 2013
- [5] **Airfoil tools:** *NACA 4 digit airfoil generator*,
Available at: <http://airfoiltools.com/airfoil/naca4digit>, Accessed on October 6, 2014
- [6] **A. Sobachin, G. Dumnov, A. Sobachkin:** *Numerical Basis of CAD-Embedded CFD*, 2014
Available at: http://www.solidworks.com/sw/docs/Flow_Basis_of_CAD_Embedded_CFD_Whitepaper.pdf, Accessed on October 6, 2014