Analysis of Pressure Coefficient Around Three Airfoils Operating at Different Reynolds Number Using CFD and XFOIL

Chapter · June 2020

DOI: 10.1007/978-981-15-4756-0_12

Parvathy Rajendran
Universiti Sains Malaysia

Analysis of Pressure Coefficient Around Three Airfoils Operating at Different Reynolds Number Using CFD and XFOIL

Aravind Seeni and Parvathy Rajendran

Abstract In this paper, CFD is used to perform analyses of 3 different airfoils at varied angles of attack. The objective is to determine a suitable CFD model and find a single validated simulation setting applicable for different airfoils. The pressure coefficient around an airfoil is analyzed. A single Reynolds number condition is assigned for testing of each airfoil over different angles of attack. The CFD results are compared with experimental and XFOIL data. The solver used is the commercially available ANSYS v16.0 CFX. From this study the following results are derived: It is found that the simulation results closely match experimental results at Reynolds number in the range of 3 million. At low Reynolds number, the CFD approach struggles to reach the higher values of pressure coefficient achieved experimentally. It is also found that XFOIL provided better results compared to CFD and also converged at a faster rate. The above results are discussed in this paper.

Keywords Airfoils · Pressure coefficient · CFD · ANSYS CFX · XFOIL · NACA0012 · NACA0015 · ClarkY

1 Introduction

There are several approaches to perform aerodynamic tests on an airfoil in fluid flow. Some of the tests are based on numerical approach such as CFD, XFOIL while others are based on experimental testing in a wind tunnel. Before the advancement of computer technology, all analyses were carried out in wind tunnel and those tests were widely reported in literature. Also a numerical method known as XFOIL was developed in the 1980s as a rapid tool to perform airfoil analysis at low Reynolds

A. Seeni · P. Rajendran (✉)
School of Aerospace Engineering, Universiti Sains Malaysia, Engineering Campus, 14300 Nibong Tebal, Penang, Malaysia
e-mail: aeparvathy@usm.my

A. Seeni
e-mail: aravindseeni@gmail.com

P. Rajendran et al. (eds.), *Proceedings of International Conference of Aerospace and Mechanical Engineering 2019*, Lecture Notes in Mechanical Engineering,
https://doi.org/10.1007/978-981-15-4756-0_12

number (*Re*) [1]. The tool uses the panel method with a linear vorticity stream-function formulation. The recent advancement in semiconductors has given rise to Computational Fluid Dynamics (CFD) as an alternate powerful tool. The use of CFD has been applied in many areas of engineering and across different fields [2–7]. CFD uses the Finite element Method for discretization of a body into smaller elements before performing numerical integration to solve for Reynolds Averaged Navier Stokes Equations in fluid dynamic analysis. The advantage of using CFD is that it can be used to perform any analysis at a reasonable computational cost. As a result, it is extensively used in aircraft structural analysis and design. For instance, it has been used in different studies for fluid dynamic analysis of an aircraft airfoil [8–12].

Since CFD follows a numerical approach the accuracy of the results is highly dependent on the modelling technique as well as simulation settings. The present research focuses on determining a suitable CFD model that can be used to analyze a variety of airfoils through a single validated simulation setting. Validation of the model will need to be performed through comparison with experimental data. For performing this, a survey of experimental results of different airfoils is undertaken that provides the pressure coefficient (*Cp*) distribution plots at different α. Different α are chosen and a comparison of experimental results with CFD and XFOIL data is subsequently performed. The modeling technique and simulation setting proposed is the output of this research. A limitation of this work is that no attempt has been made to capture the intricate flow physics, boundary layer separation and reattachment. No discussion has been made to that effect. The focus is only on obtaining closest *Cp* values to experimental results.

2 Survey of Experimental Data for Different Airfoils

2.1 Study by Gregory and O'Reilly (1970)

The study by Gregory and O'Reilly [13] is an experimental test on NACA0012 airfoil for *Re* flows of 2.88×10^6. The tests were conducted on a 4×2.7 m low speed wind tunnel at N.P.L, UK. The *Re* of flow tests were carried out in the range of 3 million i.e. 2.88×10^6 for chord lengths (c) of 0.76 m and 55 m/s flow velocity (V_∞). The results of the test were characterized by laminar separation near the leading edge without *Re*-attachment as well as turbulent boundary layer separation at the trailing edge. Undesired 3D flow behavior across the section was characterized in this 2D test. A description of the study is provided in Table 1.

Table 1 Description of study by Gregory and O'Reilly [13]

Property	Description
Dataset	Gregory and O'Reilly [13]
Location of test	Aerodynamics division, N.P.L., UK
Airfoil	NACA0012
Chord length	0.76 m
Reynolds number	2.88×10^6
Flow velocity	55 m/s

Table 2 Description of study by Marchman and Werme [14]

Property	Description
Dataset	Marchman and Werme [14]
Location of test	Virginia Polytechnic Institute and State university, USA
Airfoil	ClarkY
Chord length	0.15 m
Reynolds number	75,000
Flow velocity	–

2.2 Study by Marchman and Werme (1984)

The study of Marchman and Werme [14] focused on studying ClarkY airfoil at very low *Re*. Data were obtained on pressure distributions on the upper and lower surface of airfoil section using a 6 × 7 ft wind tunnel. While the *Re* of flow investigated was in the range of 50,000 to 200,000, 75,000 is particularly investigated for studying *Re* effects for separation. The flow turbulence level was measured at 0.05%. Flow separation was observed at $\alpha = 14°$. The study was conducted for α ranging from 0 to 28° in steps of 1° forward and back, to study hysteresis effect if present. A description of the study is provided in Table 2.

2.3 Study by Miller (2008)

The study by Miller [15] provides pressure distribution data around a NACA0015 airfoil for flow velocities of 17 m/s and *Re* of 232,940. Four α namely, 0°, 5°, 10° and 15° were investigated. The study was performed to study the lift, drag and moment characteristics of the symmetrical airfoil as well as study the hysteresis effect during stall. A description of the study is provided in Table 3.

Table 3 Description of study by Miller [15]

Property	Description
Dataset	Miller [15]
Location of test	Ohio State university, USA
Airfoil	NACA0015
Chord	0.2 m
Reynolds number	232,940
Flow velocity	17 m/s

3 Methodology

3.1 Numerical Approach

The implementation of CFD analysis is on ANSYS CFX v16.0, a general purpose CFD simulation code. The XFOIL is chosen as an alternate numerical approach which uses the Panel Method.

The airfoil coordinates are collected from standard airfoil databases. The ordinates are loaded to the software from which the points are connected together to form lines and 3D surface. A thickness of 0.01 m is assumed for enabling a 3D analysis in ANSYS.

3.2 CFD Computational Domain

The domain for analysis is sketched as a C-mesh. A spacing of 12.5c is allowed between the airfoil and the wall and 12.5c between the airfoil trailing edge and pressure outlet. This spacing is sufficient enough to prevent the development of flow affecting the result convergence. A gauge pressure of 0 is set at the outlet. The no slip condition is enabled at the airfoil surface. Similarly a zero shear condition is enabled at the outer walls.

For the grid generation, an unstructured mesh strategy is followed for grid generation of flow domain (see Fig. 1a). The region near the airfoil wall is packed with finer elements while the remaining regions consisted of coarse elements (see Fig. 1b). The region surrounding the wall is designed with wedge elements using "inflation" property in ANSYS to capture the effects associated with thin boundary layer. Thereafter, boundary conditions are applied and simulation settings are applied.

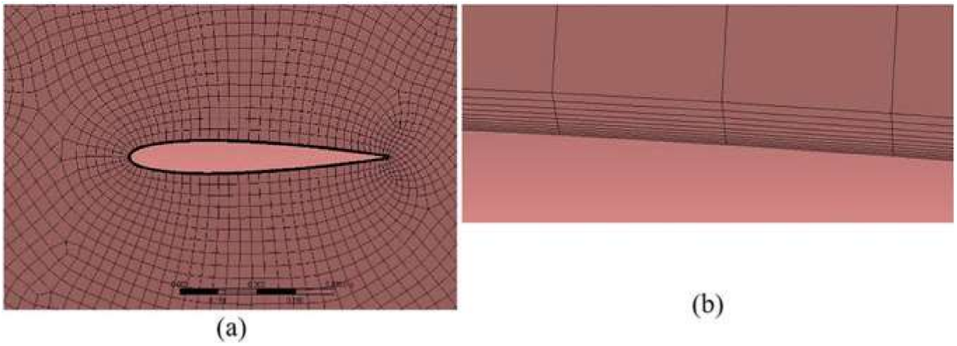

Fig. 1 **a** Unstructured mesh generation for flow domain, **b** Mesh close-up near airfoil wall for capturing viscous, thin boundary layer effects

3.3 XFOIL Domain

Similarly, the ordinates are loaded in JBLADE, an integrated XFOIL/XFLR5 interface supported by an intuitive GUI [16]. The distribution of points are "refined" further to form smooth, unobtrusive surface with the sharp curvatures at leading edge are corrected for roundness. A total of 250 points are created to form a 2D airfoil spline.

In JBLADE XFOIL, the refined points are simulated under similar conditions as suggested in experimental tests. A similar approach is followed for boundary conditions as in CFD and the results are extracted for further analysis.

3.4 Boundary Conditions and Simulation Settings

In ANSYS CFX, the simulation settings are chosen independent of any airfoil as all the simulation should be run under uniform settings. The boundary conditions are

Table 4 Domain and boundary conditions

Airfoil	NACA0012	ClarkY	NACA0015
Re	2,880,000	75,000	232,940
ρ	1.225	1.225	1.225
c	0.76	0.15	0.2032
μ	1.7894E-05	1.7894E-05	1.7894E-05
$V\infty$	55.35	7.189	16.75
Mach	0.1667	0.0217	0.0504

however varied such that velocity, chord and *Re* are maintained similar to experimental data. These are listed in Table 4. These conditions are set identical in XFOIL. A convergence criteria set at 10^{-6} is sufficient for reasonable accuracy of results.

4 Results and Discussion

4.1 Benchmark Data and Error Estimation

For establishing a comparison between numerical and experimental data, a benchmark dataset is needed. For NACA0012, data provided by Gregory and O'Reilly (1970), for ClarkY, data provided by Marchman and Werme (1984) and for NACA0015, data provided by Miller (2008) will be fixed as benchmark data. This will be assumed to have zero error and provide reference data for this work.

The datasets obtained from CFD and XFOIL are compared with benchmark data. The comparison is performed by finding the difference between obtained CFD and benchmark values for each chord length percent. These are considered as error estimates. The difference between CFD and benchmark value ($Error_1$) as well as XFOIL and benchmark value ($Error_2$) are found using the following expressions for each α as,

$$Error_1 = Cp_{CFD} - Cp_{bm} \tag{1}$$

$$Error_2 = Cp_{XFOIL} - Cp_{bm} \tag{2}$$

where Cp_{CFD}, Cp_{XFOIL} and Cp_{bm} are pressure coefficients of CFD, XFOIL and benchmark (experimental) respectively.

The *Cp* around the airfoil for different α is shown in Fig. 2 for NACA0012, Fig. 3 for ClarkY and Fig. 4 for NACA0015.

4.2 NACA0012

The error for 0°, 10° and 15° are found to be −0.1, 1.2 and 0.3 respectively (see Tables 5 and 6). From this it can be seen that the CFD values have achieved a good quality of results at low and high α. At mid α, the quality suffered due to peaks at leading edge.

In the case of XFOIL, the results are found to be very close to experimental values. There is no significant difference between experimental and XFOIL at all α and chord lengths. Thus, the values of XFOIL are found to have better quality than observed for CFD in case of NACA0012 airfoil.

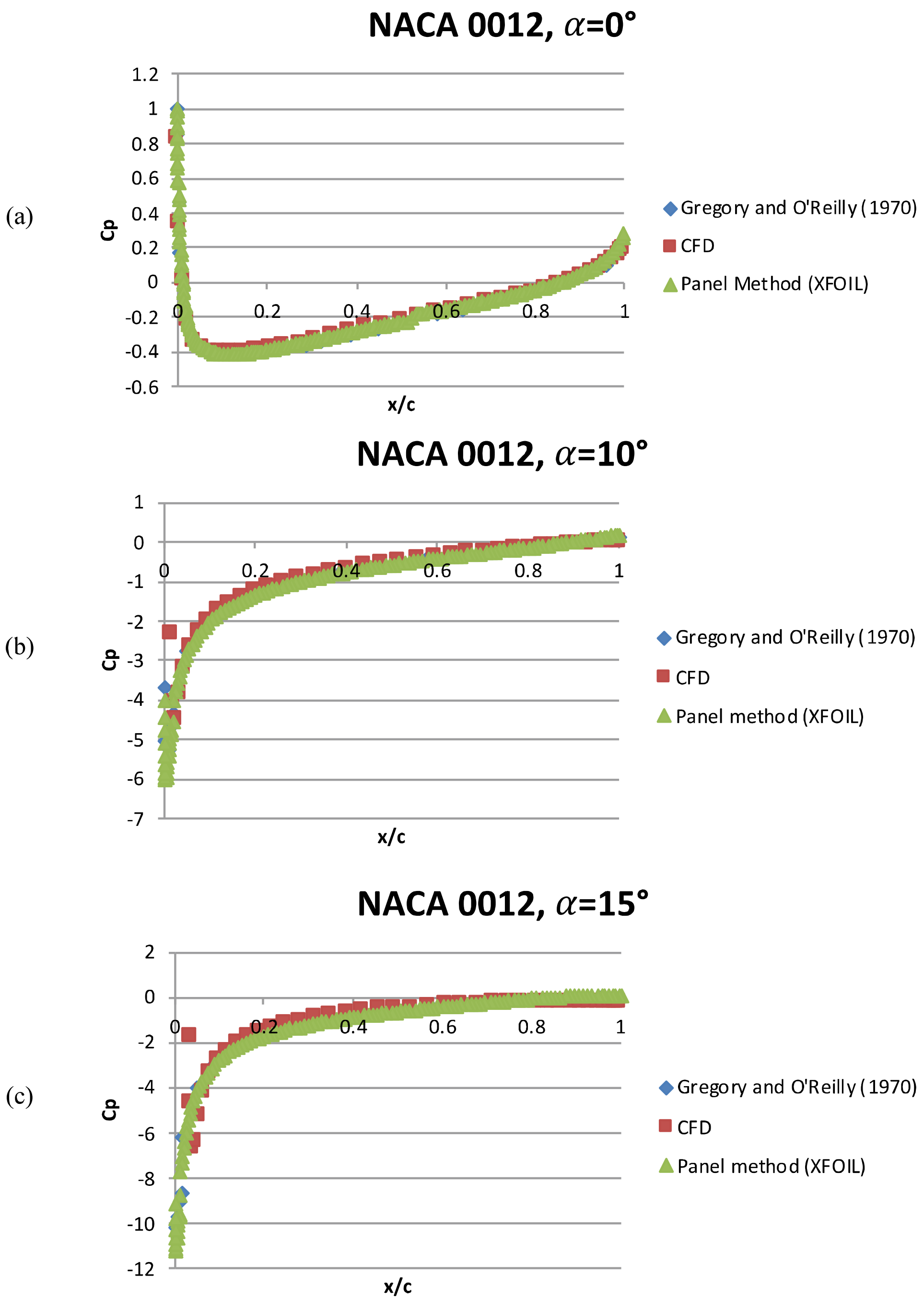

Fig. 2 Pressure coefficient plots of NACA0012 airfoil at **a** $\alpha = 0°$, **b** $\alpha = 10°$ and **c** $\alpha = 15°$

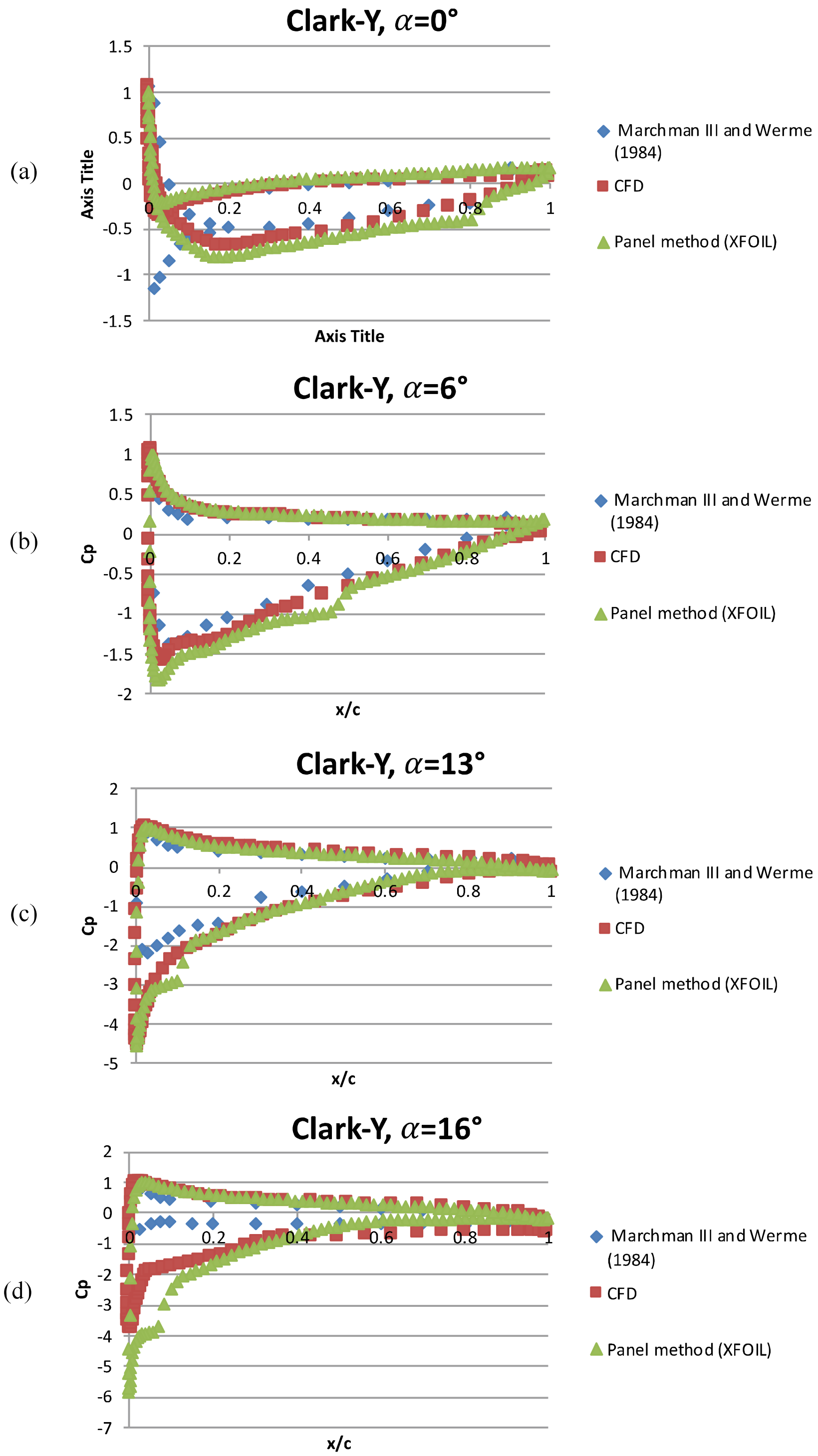

Fig. 3 Pressure coefficient plots of ClarkY airfoil at **a** $\alpha = 0°$, **b** $\alpha = 6°$, **c** $\alpha = 13°$ and **d** $\alpha = 16°$

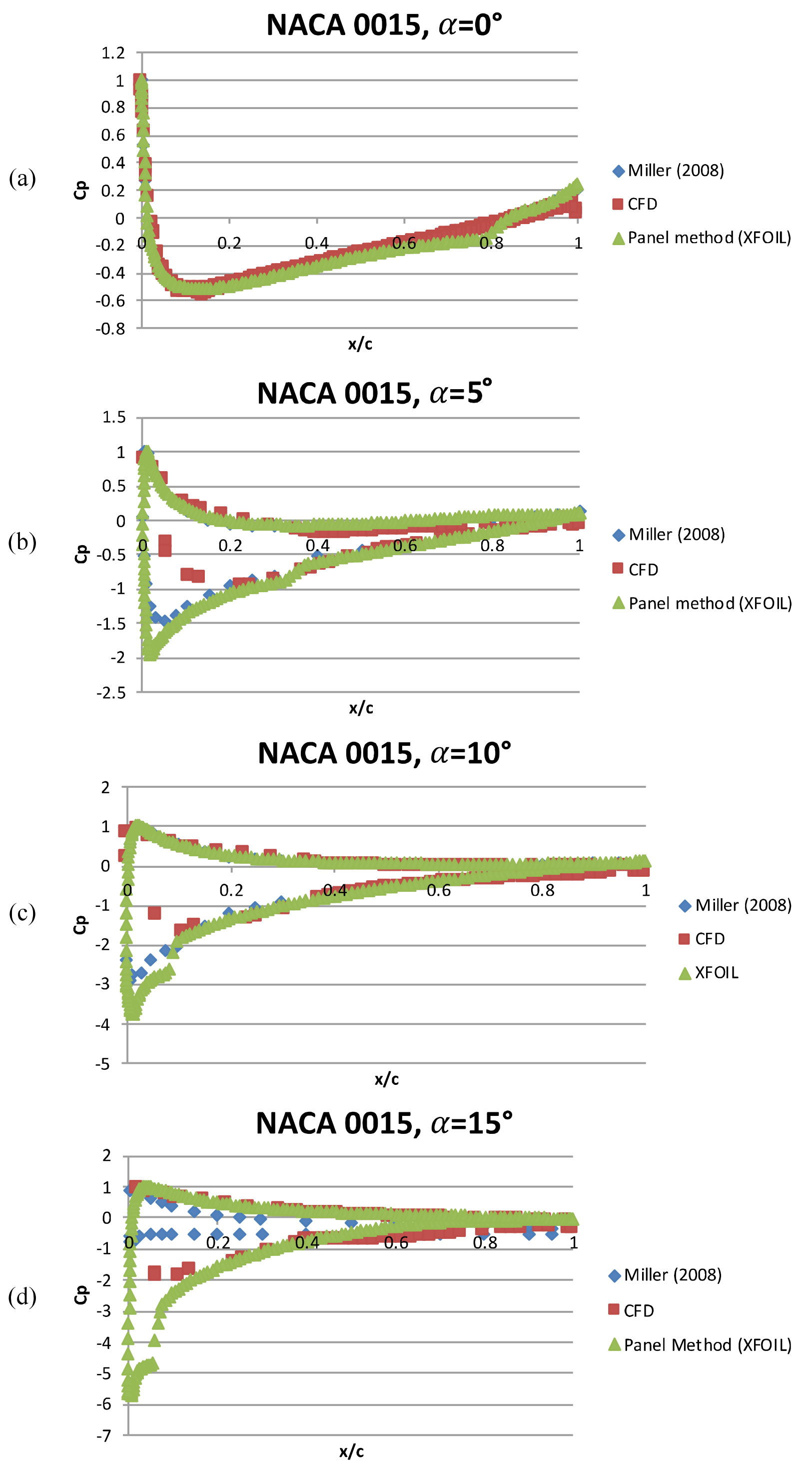

Fig. 4 NACA0015 simulation and experimental results comparison. **a** $\alpha = 0°$, **b** $\alpha = 5°$, **c** $\alpha = 10°$ and **d** $\alpha = 15°$

Table 5 Error estimates for different angles of attack from 0º to 10º

Airfoil	$\alpha = 0°$		$\alpha = 5°$		$\alpha = 6°$		$\alpha = 10°$	
	$Error_1$	$Error_2$	$Error_1$	$Error_2$	$Error_1$	$Error_2$	$Error_1$	$Error_2$
NACA0012	−0.1	−0.1	–	–	–	–	1.2	0.1
ClarkY	0	−0.1	–	–	−0.1	−0.1	–	–
NACA0015	0	0	0.5	0.1	–	–	0.1	0

Table 6 Error estimates for different angles of attack from 13º to 16º

Airfoil	$\alpha = 13°$		$\alpha = 15°$		$\alpha = 16°$	
	$Error_1$	$Error_2$	$Error_1$	$Error_2$	$Error_1$	$Error_2$
NACA0012	–	–	0.3	0	–	–
ClarkY	−0.3	−0.1	–	–	−0.5	−0.3
NACA0015	–	–	0.5	−0.1	–	–

4.3 ClarkY

The error estimate shows that all the CFD values have performed uniformly similar to all α,—0°, 6°, 12° and 14° (see Tables 5 and 6). At all α, the leading edge suction peak occurs. The maximum error is found to be at higher α which implies that the results are obtained as expected. At 14° α, the maximum error of −0.5 is obtained.

The error in XFOIL is found to be lower than CFD with no leading edge peaks. The maximum error observed is −0.3 for 14° α. For other α, the error is observed to be relatively less at −0.1 which explains the better results output of XFOIL.

4.4 NACA0015

The error estimate in the case of NACA0115 for CFD is close to 0 at 0° α. Leading edge peaks are observed at higher α the overall error average is found to be maximum for 5° and 15° and lowest at 0° (see Tables 5 and 6).

The error results obtained show than XFOIL has produced better results compared to CFD for all α.

5 Conclusion

A comparative analysis between CFD, XFOIL and Experimental has been carried out. The results show that out of two numerical methods, XFOIL had closer predictions to benchmark values for all 3 airfoils. It was also found that the CFD simulation in airfoils is applicable only for *Re* of at least 75,000. The ANSYS CFX solver with simulation settings proposed has yielded satisfactory results.

References

1. Drela, M (1989) XFOIL: an analysis and design system for low Reynolds number airfoils. In: Conference on low reynolds number airfoil aerodynamics, vol 54, pp 1–12
2. Addepalli SK, Mallikarjuna JM (2016) Parametric analysis of a 4-stroke GDI engine using CFD. Alexandria Eng J 57(1):23–34
3. Babich F, Cook M, Loveday D, Rawal R, Shukla Y (2017) Transient three-dimensional CFD modelling of ceiling fans. Build Environ 123:37–49
4. Bottarelli M, Bortoloni M, Zannoni G, Allen R, Cherry N (2017) CFD analysis of roof tile coverings. Energy 137:391–398
5. Močilan M, Žmindák M, Pecháč P, Weis P (2017) CFD simulation of hydraulic tank. Proc Eng 192:609–614
6. Prachař A, Hospodář P, Vrchota P (2018) Gust alleviation of aeroelastic aircraft using CFD simulation. Transp Res Proc 29:366–375
7. Vittorio C, Aldo F, Karim AS, Vincenzo B, Emanuele R, Marco M (2018) Preliminary transonic CFD analyses of a PrandtlPlane transport aircraft. Transp Res Proc 29:82–91
8. Nordanger K, Holdahl R, Kvamsdal T, Kvarving AM, Rasheed A (2015) Simulation of airflow past a 2D NACA0015 airfoil using an isogeometric incompressible Navier-Stokes solver with the Spalart-Allmaras turbulence model. Comput Methods Appl Mech Eng 290:183–208
9. Nordanger K, Holdahl R, Kvarving AM, Rasheed A, Kvamsdal T (2015) Implementation and comparison of three isogeometric Navier-Stokes solvers applied to simulation of flow past a fixed 2D NACA0012 airfoil at high Reynolds number. Comput Methods Appl Mech Eng 284:664–688
10. Bragg MB, Broeren AP, Blumenthal LA (2005) Iced-airfoil aerodynamics. Prog Aerosp Sci 41(5):323–362
11. Morgado J, Vizinho R, Silvestre MAR, Páscoa JC (2016) XFOIL vs CFD performance predictions for high lift low Reynolds number airfoils. Aerosp Sci Technol 52:207–214
12. Kapsalis PS, Voutsinas S, Vlachos NS (2016) Comparing the effect of three transition models on the CFD predictions of a NACA0012 airfoil aerodynamics. J Wind Eng Ind Aerodyn 157:158–170
13. Gregory N, O'Reilly CL (1970) Low-speed aerodynamic characteristics of NACA 0012 aerofoil section, including the effects of upper-surface roughness simulating hoar frost. Reports Memo 3726(3726):35
14. Marchman JF, Werme TD (1984) ClarkY airfoil performance at low reynolds numbers. In: AIAA 22nd aerospace sciences meeting
15. Miller SD (2008) Lift, dragand moment of a NACA 0015 airfoil, p 25 (2008)
16. Morgado J, Silvestre MÂR, Páscoa JC (2015) Validation of new formulations for propeller analysis. J Propuls Power 31(1):467–477

CFD Analysis of a Novel Propeller Design Operating at Low Reynolds Number

Aravind Seeni and Parvathy Rajendran

Abstract The aerodynamic performance of a propeller is characterized by the thrust produced, torque and efficiency. In this paper, novel blade shapes are tested using the technique Computational Fluid Dynamics (CFD). The blade shapes are based on a passive slotted design in which slots are present on the suction side i.e. on the upstream front side of the blade. Two slotted design models are tested using the commercial CFD solver ANSYS Fluent. The slotted designs are identical in terms of slot geometry (0.1 mm width, 0.2 mm depth) but differ in the location of slot along chord from leading edge (0.206c and 0.382c). In order to test the modified designs, a validation study of the baseline model of a propeller is conducted. Then slotted designs comprising two models are tested and compared to baseline design. The propeller model considered is the APC10x7 Slow Flyer and the Reynolds number of flow analyzed is approximately 68,500 (estimated at 75% radial distance) The results showed that the presence of slots have altered the performance. The presence of slots has reduced the thrust performance for the entire operational range of advance ratios. However, the power performance has increased due to decrease in torque or rotational resistance of the propeller. The efficiency is thereby increased for one model for a specific range of advance ratios. In the case of another model, for most advance ratios the efficiency was found to be decreased.

Keywords Slotted design · Propeller performance · Computational fluid dynamics · Validation · Thrust · Power · Efficiency

A. Seeni · P. Rajendran (✉)
School of Aerospace Engineering, Universiti Sains Malaysia, Engineering Campus, 14300 Nibong Tebal, Penang, Malaysia
e-mail: aeparvathy@usm.my

A. Seeni
e-mail: aravindseeni@gmail.com

P. Rajendran et al. (eds.), *Proceedings of International Conference of Aerospace and Mechanical Engineering 2019*, Lecture Notes in Mechanical Engineering,
https://doi.org/10.1007/978-981-15-4756-0_13