

---

# Low Wind Speed Wind Turbine Blade Design for Thailand

---

Natchanon Suppaadirek<sup>1</sup>, Noppakoon Saengsuwan<sup>2</sup>, Sirichai Tammaruckwattana<sup>3</sup>

<sup>1,2,3</sup> Faculty of Engineering, King Mongkut's Institute of Technology Ladkrabang, Bangkok, Thailand.

Corresponding author; E-mail address: 62601186@kmitl.ac.th<sup>1</sup>, 62601255@kmitl.ac.th<sup>2</sup>, sirichai.ta@kmitl.ac.th<sup>3</sup>

## Abstract.

Renewable energy is a reason why this study has been chosen. In this paper focused on an analysis of the characteristics of the wind when the wind turbine blade is encountered using a commercial fluid dynamics solver such as Fluent by selecting the impact angle of 5, -5, 10, 15, 20, -10, -15 and -20 degrees. Aerodynamic characteristics such as lift coefficient, drag coefficient, lift to drag ratio has been evaluated in this study. Fluent simulation results will be compared with the result of the milling airfoil tested in wind tunnel. Our goal is to a construction of wind turbine in low wind speed country like Thailand

**Keywords.** Airfoil, Wind turbine blade, lift coefficient, Wind tunnel experiment

## 1. INTRODUCTION

Nowadays, the increase in energy requirement have been observed due to increasing in technology and industrialization. Energy requirement and development level of the countries have shown parallelism. However, except for renewable energy methods, the other methods in the generation of energy causes to increase greenhouse gas and also, increase global warming. Wind energy is one of the renewable energy sources and wind turbines are used to generate electrical energy by using the kinetic energy of wind.

One of the most important part of wind turbines is their aerodynamic effectiveness, the base of which is the design of the airfoils forming the wind turbine blade shape.

As a major component of wind turbine generator system, blade has been paid more and more attention to, and its design and manufacture have become a kind of hot technology problem [1~3] in wind power industry. The shape, airfoil, materials, number of blades, structure and processing technology of the blade have been deeply studied both at home and abroad [4~7]. Aerodynamic shape which affects the strengths and weaknesses of the whole unit performance is an important factor. On the basis of the previous study [8], by improved the current optimal design method.

In this paper proposes the wind turbine blade shape design and the use of Ansys Fluent. The simulation results obtained from Fluent are compared to the data from real wind tunnel. Airfoil data presented here, were obtained from airfoil tools.

## 2. AIRFOIL PROFILE DESIGN

An airfoil is the cross-sectional shape of a wind turbine blade, its body moving through a fluid produces an aerodynamic force. The wind come contact with lead edge part of airfoil

Airfoil configuration has been mention in [5]. The component of aerodynamic force perpendicular to the direction of motion is called lift and the component parallel to the direction of motion is called drag, also the angle between the direction of the apparent or relative wind and the chord line of the blade is called Angle of attack.

In this paper NACA 4412 airfoil is considered as base airfoils. NACA 4412 wing profile is a generic geometry frequently used as an industrial and academic test case to validate simulation and experimental methods and it has low drag and increased lift forces at relatively low wind speeds which is suitable for our research that focus on low wind speed turbine.

NACA MPXX, the first digit expresses the maximum camber divided by 100, the second digit gives the position of the maximum camber divided by 10, and the last two digits gives the thickness divided by 100. Thus 4412 has a maximum camber of 4% of chord located at 40% chord back from the leading edge and it thickness is 12% [7].

NACA 4412 Dimension taken from the NACA four digits generator in airfoiltool. After obtained the coordinate data XY from airfoil NACA 4412, the next step is to convert it to Text file using Microsoft Excel in XY coordinate.

The XY coordinate data of NACA4412 are used to draw an airfoil in Fluent ANSYS.

## 3. AIRFOIL SIMULATION

In this study using the method of Fluent, where the process sequence from Modeling to produce images contours of wind velocity can be described as follows:

- Run ANSYS Workbench in the Fluid Flow (Fluent) category. The first thing to do is to insert XY coordinate data from text file into Geometry block (Design modeler geometry) and draw airfoil from the coordinate point.
- Draw boundary to make a free-stream conditions.
- After finished the geometry, go to meshing.
- After meshing finished, the model parts are generated as Inlet, Outlet, walls and as airfoil surfaces.
- Thereafter the model is run into Fluent version 17.2 by providing airflow velocity rates, external air pressure, material type, experimental temperature, air viscosity, and air density

- After the data are complete, then Fluent can do running by entering the amount of data iteration.
- Create a picture of velocity contour.
- Input angle of attack, lift and drag coefficient equation in parameter box.
- Then the Fluent will calculate lift and drag coefficient base on angle of attack and velocity.

After finishing the simulation process, the program can change into variety velocity image depended on angle of attack that been put into before the calculate sequence, also can calculate lot of aerodynamic data conditional on parameter input.

From our research and another reference that been use in this paper, we can group them into three methods. First method shown wind or pressure behavior upon airfoil which [1] use CATIA to create model and STAR-CCM to simulate. Ananth S Sharma [3] using ICEM-CFD software to mesh model and transfer it into Fluent. Second method is Q-blade technique which [5],[8] focus on obtain the aerodynamics data rather than contour image. Third method use BEM equation to calculated aerodynamics data base on NACA 4412 [7]. Three method detail has shown in Figure. 1.

Method	Paper Reference	Program design	Program Simulation	Lift	Drag	Cl	Cd	Cl/Cd	Cp	Wind Tunnel
Simulation with velocity or pressure contour and collected data	Our	Design modeler	Fluent	✓	✓	✓	✓	✓	✗	✓
	[1]	CATIA	STAR-CCM	✓	✓	✓	✓	✓	✗	✓
	[3]	ICEM CFD	Fluent	✗	✗	✗	✗	✗	✗	✗
Simulation with airfoil image and collected data	[5]	Q-blade	Q-blade	✓	✓	✓	✓	✓	✗	✗
	[8]	Q-blade	Q-blade	✓	✓	✓	✓	✓	✓	✗
Calculated from equation	[7]	BEM	—	✓	✓	✓	✓	✓	✓	✗

**Figure 1** Comparison of Simulation Method

In this paper, we use ANSYS fluent to simulate airfoil to collect velocity contour and aerodynamic data such as lift and drag. If we use STAR-CCM+ and ANSYS Fluent to simulate the same thing in same condition, the results reached grid independent solutions with the same mesh size, and the accuracy of the predicted results was similar but with

calculated airflow, ANSYS Fluent performed slightly better, and its userdefined functions were more user-friendly and suitable for low spec pc or laptop[9], so that why this paper using Fluent instead of STAR-CCM. Fluent can install the BEM equation to the parameter function and some of the equations are already exist in program. It simpler to use simulation program to generate data rather than calculate the data by hand and simulation is more visualize

In aerodynamic have aerodynamic force called lift and drag, so we also observed and collected lift and drag data to study their behavior compare to vary of velocity and angle of attack. Coefficient  $C_l$  and drag coefficient  $C_d$  of airfoil are calculated with Fluent with (1) and (2).

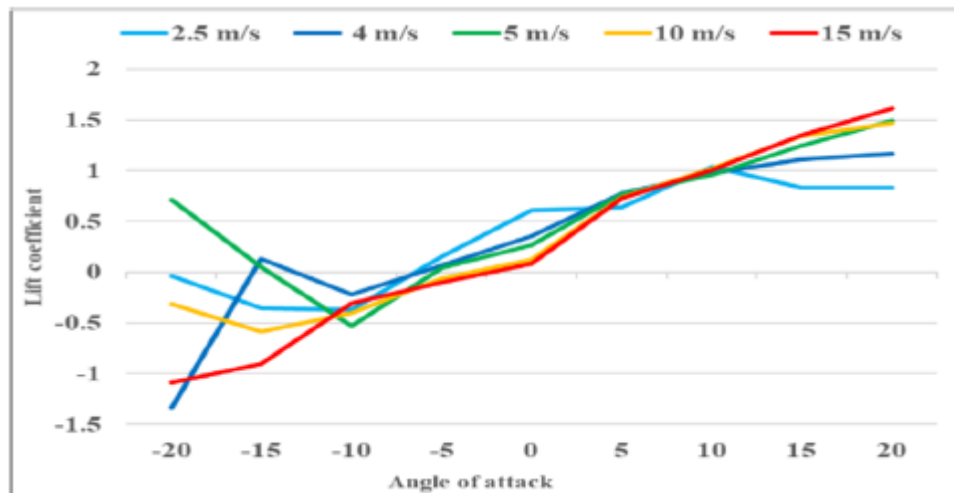
$$c_l = \frac{2L}{\rho v^2 A} \quad (1)$$

$$c_d = \frac{2D}{\rho v^2 A} \quad (2)$$

Where,  $\rho$ :air density L : Lift D : Drag A : area v :velocity

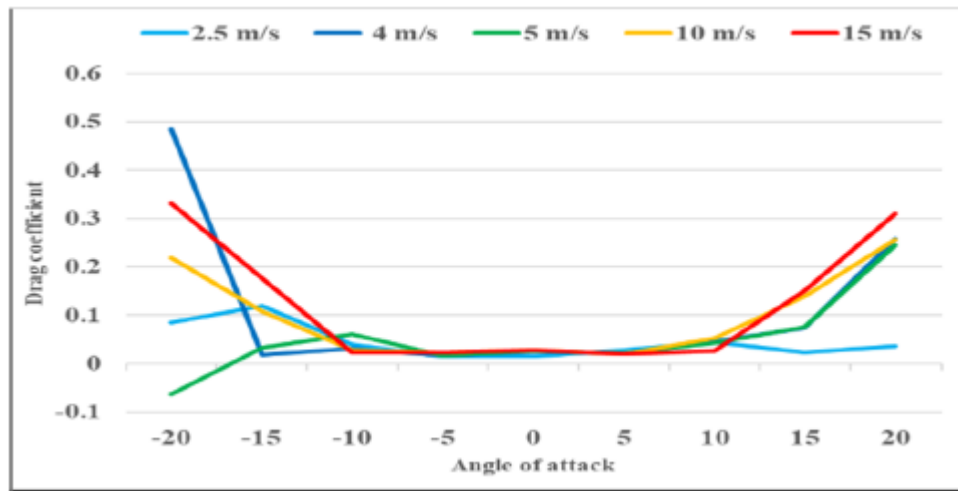
Base on Fig. 2 as angle of attack increases, the value of lift coefficient also increases to a certain point and then started to decrease. Also, the value of lift coefficient also increases when velocity increases when tested in the same angle of attack condition.

We can assume that lift will continue to rise with angle of attack until it reaches maximum value and then started to fall.



**Figure 2** Lift coefficient

Another aerodynamics behavior that also contributes in producing good wind turbine is drag coefficient. The result of drag coefficient is presented in Fig. 3. It was show that, the drag coefficient is increased as angle of attack or velocity with an exception when drag reach its maximum value and start to fall down. In actual of wind turbine blade development, a resistant drag force which opposes the motion of the blade must also be minimum, so the main point is to minimize the drag force in order to make the lift more dominance.

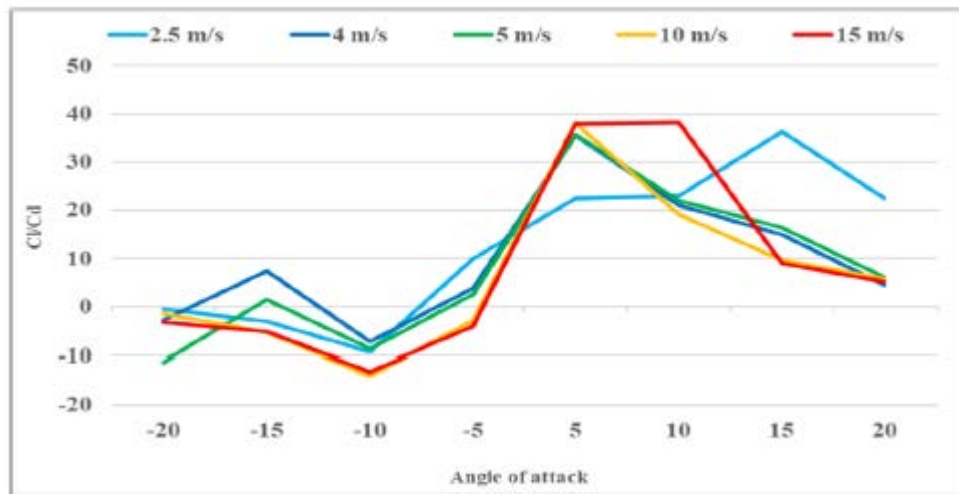


**Figure 3** Drag coefficient

This result also relatively conflict with requirement of maximizing the lift coefficient to compensate the drag coefficient produced because lift coefficient and drag coefficient increases in the same way

In this case, we have to calculate lift to drag ratio to determine how much the dominance of lift coefficient compare to drag coefficient in each angle of attack .We can calculate lift to drag ratio with (3). The results will specify which angle of attack had to be choose as an appropriate point. High lift to drag ratio means that drag is less effective on that airfoil. Fig. 4 show the results of lift to drag ratio with the vary of velocity and angle of attack in graph.

$$\text{Lift to drag ratio} = \frac{\text{Coefficient of Lift}}{\text{Coefficient of Drag}} = \frac{C_L}{C_D} \quad (3)$$



**Figure 4** Graph of Lift to Drag Ratio

As you can see in Fig. 4 the maximum lift to drag ratio in each wind speed are not the same. For 15m/s the maximum lift to drag ratio is at 10°, for 5m/s is 5° and similar to 4m/s. We also can be observed that lift to drag ratio start to decrease when it reached maximum value such as 15 m/s decrease at angle of attack of 10, 4 m/s at 5.

The example of the simulation results the aspect of velocity of NACA 4412 airfoil under different angles of attack is shown in the following Fig. 5 (a, b, c, d). The velocity coefficient varied largely under different attack angle It can be seen that in positive angles of attack, the velocity on the leading-edge surface was high and another side that didn't make contact with wind was low. In negative angles of attack, the wind behaves the same but with an opposite direction

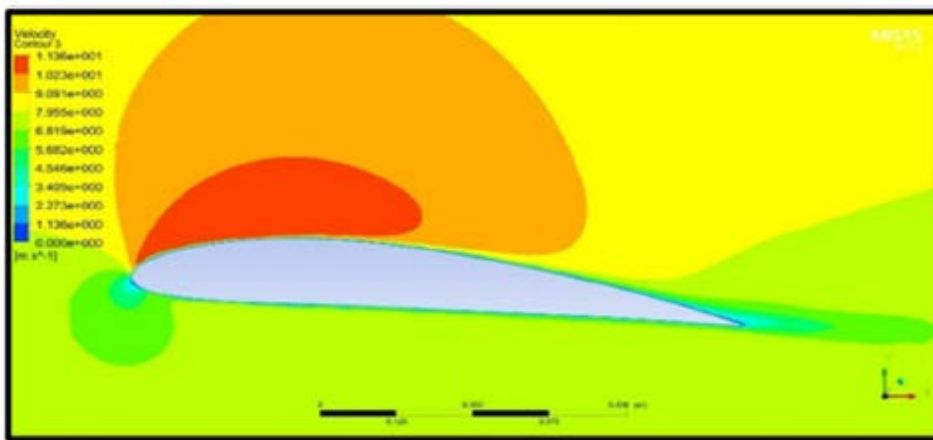
#### 4. EXPERIMENTAL SYSTEM

The 3D printing model of an airfoil will be used to test in wind tunnel. Fig. 5 and 6 show the simulation results of variety angle of attack and the experimental results of variety angle of attack. for the low wind speed wind turbine blade design are implemented by using Ansys designmodeler/Fluent. The simulation models consider the wind velocity, the wind turbine blade shape. The all schemes were simulated for the same change of wind velocity. The simulation results verify the fundamental performance of the wind turbine blade shape

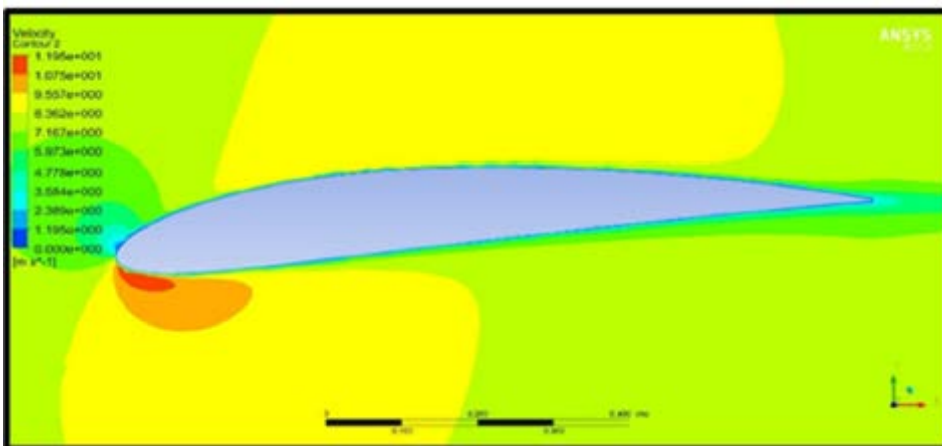
#### 5. CONCLUSION

This paper work was proceeded to study the aerodynamics behavior of the chosen airfoil from Fluent simulation. From this research can be concluded as below.

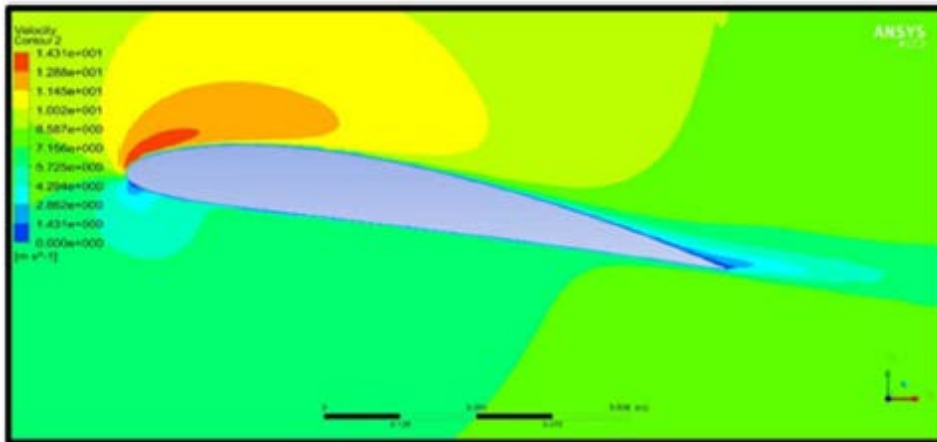
- 1 Wind speed and angle of attack have influenced with Lift and Drag coefficient. Higher velocity and higher angle of attack increased Lift and Drag coefficient but the value will start to decrease when reach their maximum value.
- 2 Maximum lift to drag ratio is different in each wind speed and show sign of decrease when it reaches maximum value
- 3 It unsuitable to use angle of attack at negative degree when you want high lift to drag ratio.
- 4 The design for wind blade made from NACA 4412 used in low wind speed zone such as Thailand should be use at angle of attack between 5 to 15



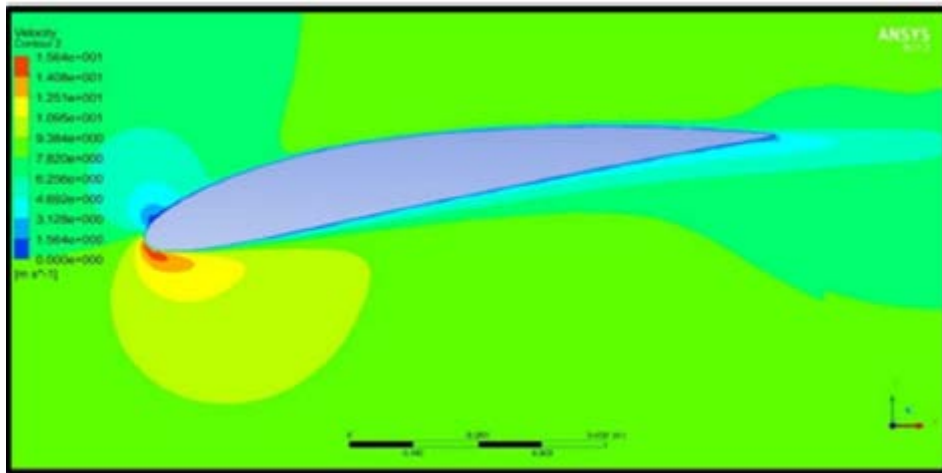
(a) Angle of Attack  $5^\circ$



(b) Angle of Attack  $-5^\circ$



(c) Angle of Attack 10°



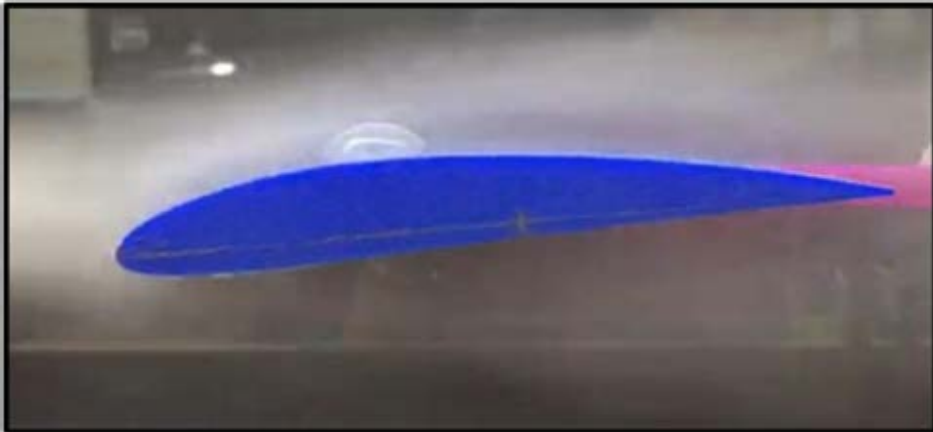
(d) Angle of Attack -10°

**Figure 5** Simulation results for variety angle of attack

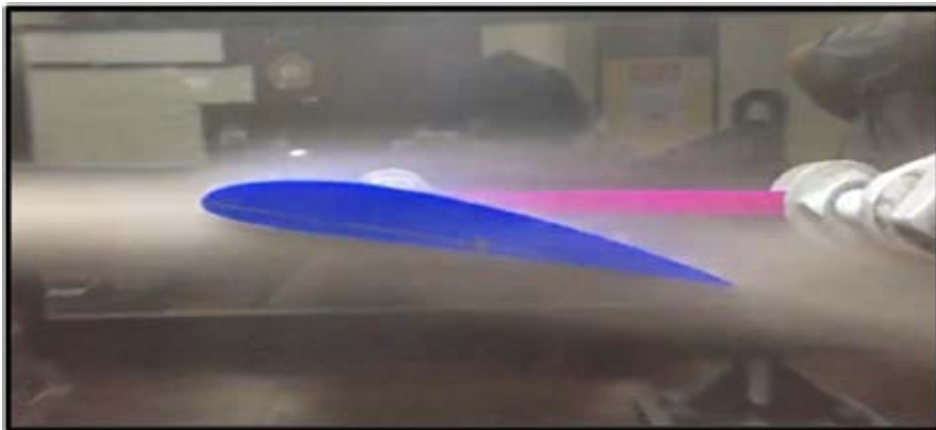




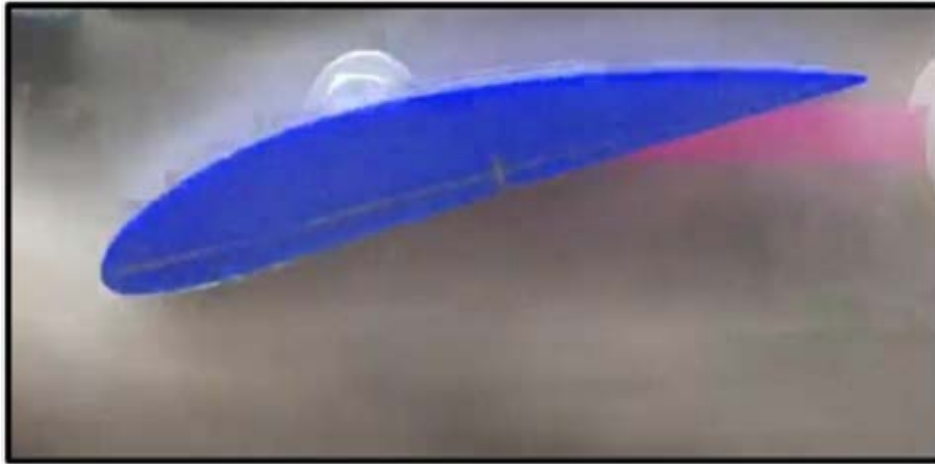
(a) Angle of Attack  $5^\circ$



(b) Angle of Attack  $-5^\circ$



(c) Angle of Attack  $10^\circ$



(d) Angle of Attack  $-10^\circ$

**Figure 6** Experimental results for variety angle of attack

## 6. REFERENCES

- [1] M.H.M. Noh, A.H.A. Hamid, H. Rashid, W. Wisnoe and M.S. Nasir, "Wind Tunnel Experiment for Low Wind Speed Wind Turbine Blade," *Mechanics and Materials*, Vols. 110-116 ,pp 1589-1593, 2012.
- [2] P.J. Schubel and R.J. Crossley, "Wind Turbine Blade Design," *Energies*, Vol. 5, pp. 3425-3449, 2012.
- [3] A.S. Sharma, Sudhakar S., Swathijayakumar and B.S.A. Kumar, "Investigation of Pressure Contours and Velocity Vectors of NACA 0015 in Comparison with Optimized NACA 0015 Using Gurney Flap," *International Journal of Mechanical And Production Engineering*, Vol. 3, Issue 9, 2015.
- [4] M.F. Hidayat and Y. Nofendri, "Aerodynamic Study Airfoil NACA 0013 with ANSYS Fluent," *Article*, 2019.
- [5] M.R. Birajdar and S.A. Kale, "Effect of Leading Edge Radius and Blending Distance from Leading Edge on the Aerodynamic Performance of Small Wind Turbine Blade Airfoils," *International Journal of Energy and Power Engineering*, Vol. 4(5-1), pp. 54-58, 2015.
- [6] X. Baoqing and T. De, "Simulation and Test of the Blade Models Output Characteristics of Wind Turbine," *Energy Procedia* 17, pp. 1201-1208, 2012.
- [7] S.A. Kale and R.M. Varma, "Aerodynamic Design of a Horizontal Axis Micro Wind Turbine Blade Using NACA 4412 Profile," *International Journal of Renewable Energy Research*, Vol.4, No.1, pp. 69-72, 2014.

- [8] P.D Adb. Aziz, A.K.R. Mohamad, F.Z. Hamidon, N. Mohamad, N. Salleh and N.M. Yunus, "A Simulation Study on Airfoils Using VAWT Design for Low Wind Speed Application," International Conference on Engineering Technology and Technopreneuship, Vol.4, pp. 105-109, 2014
- [9] Y. Zou<sup>1</sup>, X.Zhao<sup>1</sup> and Q. Chen, "Comparison of STAR-CCM+ and ANSYS Fluent for Simulating Indoor Airflows", Building Simulation, Vol.11(1), pp 165-174, 2018
- [10] E. Koc, O. Gunel and T. Yavuz, "Comparison of Qblade and CFD results for small-scale horizontal axis wind turbine analysis," IEEE International Conference on Renewable Energy Research and Applications, 2016