



DESIGNING A WIND TURBINE BLADE: AERODYNAMIC DESIGN

BEng Energy & Environmental (Level 6)

Submitted by: Omaira Hernandez Mosegue

Submitted to: Kelvin Lake

Date: 11th January 2020

Module: Structural & Fluid Analysis

Module code: ACMA6010

1. TABLE OF CONTENTS

1. TABLE OF CONTENTS	1
2. INTRODUCTION	2
3. FLUID-STRUCTURE INTERACTION TECHNIQUES	2
○ 3.1 WIND TUNNELS	2
○ 3.2 COMPUTATIONAL FLUID DYNAMICS	3
4. FLUID DYNAMICS	3
5. AIRFOILS	3
6. REYNOLDS NUMBER	4
7. BOUNDARY LAYERS FLOW	5
8. ANGLES FORMED BETWEEN THE BLADE AND WIND FORCES	6
○ 8.1 FLOW ANGLE	7
○ 8.2 ANGLE OF ATTACK	7
○ 8.3 PITCH ANGLE: TWIST ANGLE AND GLOBAL PITCH ANGLE	7
9. CONCLUSION	8
10. REFERENCES	10

2. INTRODUCTION

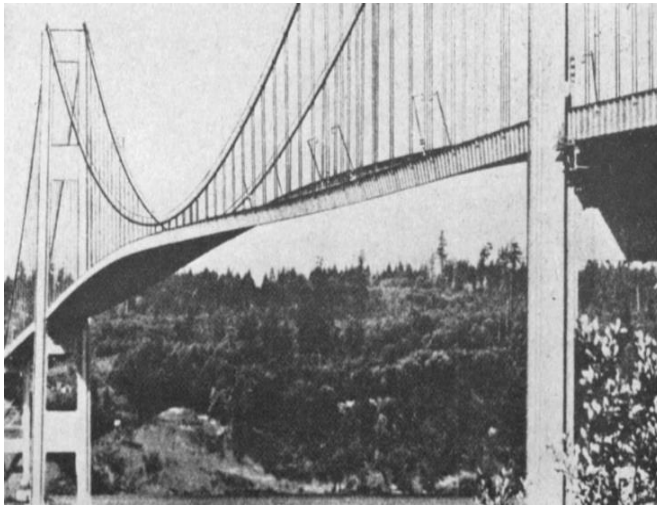


Figure 1. Photograph of the Tacoma Narrows bridge before collapsing (Païdoussis, Michael P, et al. 2010, p.19)

The demand for engineering components that would be subject to vibration behaviour and fatigue life due to various reasons, such as rough operating conditions or weight reduction to achieve high performance, has drastically increased. In [Figure 1](#), a photograph of the Tacoma Narrows bridge before collapsing can be appreciated. This structure collapsed due to flow separation that caused aeroelastic flutter, mostly known as the uncontained vibrations that can lead to the breakage of a structure or solid body in contact with fluid (Païdoussis, Michael P, et al. 2010). Therefore, Fluid-Structure Interaction (FSI) is of vital importance during the engineering

design phase. FSI analyses the contact of a fluid with structures that are subjected to the fluid flow's forces and vibrations resulting from high-velocity turbulent fluids. A clear fluid and structural example is wind turbines, in which air is the flow physic and the wind turbine the solid surface. These structures are exposed to vibrations that could lead to fatigue or wear in the long term or even some severe damage in the short term (Païdoussis, et al. 2010).

The main aim of this assignment is to evaluate different parameters of great importance in a Fluid-Structure Interaction analysis of wind turbine blades which are exposed to vibrations and fatigue. Moreover, the blades to analyse in this document rotate around an axis that is perpendicular to the incoming flow, thus the turbine is a crossflow wind turbine rather than a axial-flow turbine.

3. FLUID-STRUCTURE INTERACTION TECHNIQUES

To achieve the best efficiency, control, noise and aesthetic of components and structures, evaluation techniques in the design phase of the manufacturing process are required. Two of the most common techniques used in numerous engineering projects to simulate the effect of wind on solid surfaces are: Wind tunnels and Computational Fluid Dynamics (CFD).

3.1 WIND TUNNELS

Wind tunnels is the most important tool in aerodynamics simulation. This method consists of hollow tubes that contain a test section in which the design model is placed. Subsequently, numerous powerful fans simulate fluid flow while the model is connected to different sensors that measure various phenomena such as: torque, vertical or axial forces, rotor rotating speed in the case of an aircraft or a wind turbine, and a brake system. Additionally, the collected data for pressure and temperature, as well as the velocities obtained from using Bernoulli's equation, are assembled, and interpreted by scientists who will use the information to optimise the object. However, experimental flow conditions might be affected due to limitation size of the wind tunnel test. Consequently, aerodynamics characteristics such as Reynolds number might be altered (Chen, T.Y. and Liou, L.R., 2011)

3.2 COMPUTATIONAL FLUID DYNAMICS

Computational Fluid Dynamics is a powerful tool to improve the design phase and optimisation of engineering objects and structures in contact with fluids. Computers are used to simulate fluid flow over digital models by using partial different equations that express the three basic principles of fluids: mass conservation, energy conservation and $F=ma$ (Newton's second law), which means that the force acting on a body is equal to its mass and acceleration. Although this technique provides specific answers by a variation of the parameters of a given problem, the necessary physic properties of the fluid are not always included (Wendt, J.F. ed., 2008).

4. FLUID DYNAMICS

Fluid dynamics is the study of fluids (liquids and gases) motion and the effects of the fluids upon the boundaries, usually solid surfaces, or other liquids. This study is so important as nearly everything on earth is either a fluid or almost a liquid or moves within it. In addition to the given information about fluids, it should be noted that fluids are known to have incapability to maintain shear stress, instead, the shear stress will cause fluid motion and continuous deformation. To analyse the motion of a fluid, the different properties at every point must be considered, which are named as velocity, pressure and there could also be changes in density. However, there are another important transport properties known as viscosity, term used to define the fluid's resistance to flow caused by a shearing stress, thermal conductivity and mass diffusivity (McDonough, J.M., 2009)

Fluid mechanics can be classified in various types, from simple mechanics such as linear or laminar to more complex ones that could cause numerous problems and therefore the most important ones to analyse which vary from unsteady, non-uniform or turbulent, among others (McDonough, J.M., 2009).

5. AIRFOILS

To understand some of the parameters that will be named during this document, it is of vital importance to comprehend the design of an airfoil.

Over most of the length of a wind turbine blade, the blade cross-section is an airfoil, which is a curved shape design that enables upcoming air to create a lift force that rotates the blades and therefore creates kinetic energy that would be converted into electricity by the turbine's mechanical equipment. The flow encounters the blade by the rounded front edge named leading edge, while the trailing edge is the edge of the blade where flow leaves. Both edges are joined by a straight chord line that cuts the airfoil into an upper and lower surface. On the other hand, the maximum distance between the upper and the lower surfaces, which is measured perpendicular to the chord line, is called the thickness. Moreover, the locus of points considered as the maximum distance between both edges of the blade which also measures the curvature of the blade is named as camber line (El Mouhsine, et al. 2018). The mentioned terms can be appreciated in Figure 2.

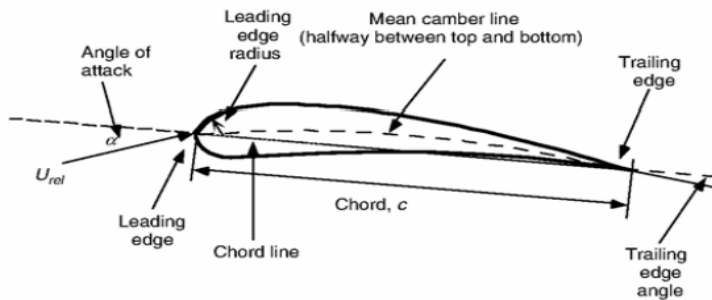


Figure 2. Sketch of an airfoil (El Mouhsine, et al. 2018, p.2054)

6. REYNOLDS NUMBER

The effect of the Reynolds number on a topic such as wind turbines is extremely important from an engineering standpoint. Nowadays, larger wind turbine blades are essential as they cover a wider area and therefore the total energy production is increased. However, the increasing Reynolds number (above 10 million) that the blades would be subjected to is a challenge to the engineering design (Pires, O, 2016).

As E. Shashi Menon (2015) stated in Transmission Pipeline Calculations and Simulations Manual, the Reynolds numbers is a parameter which helps obtaining a flow pattern estimation in various fluid flow situations. Besides the fact that calculating the dependence of a parameter on the Reynolds formula is almost impossible due to the couple behaviour as well as the non-linearity of flow equations, simulations and experiments are carried out to understand the governing equations (Miller, M.A.M., 2018). Moreover, the principal reason why this mathematical construct is so important for the flow around a wind turbine is because it determines the flow regime around the blade. The Reynolds number affects the fluid around by changing the movement of the fluid's particles, so the flow would follow the blade surface at low Reynolds number or leads to turbulence, separation, and extra drag at high Reynolds number. Therefore, it can be used to achieve a better lift-to-drag ratio as well as improving the angle of attack, critical parameters for wind turbine blades to spin and for losses over the design operating range to be significantly reduced (Miller, M.A.M., 2018).

In this section, an estimation example of the Reynolds number of the fluid around a wind turbine blade is estimated.

The Reynolds number is mathematically defined as (Rehm, B., et al. 2008):

$$R = \frac{\rho V D}{\mu} = \frac{V D}{\nu}$$

where:

- ρ is the fluid density (kg/m³)
- V is the flow velocity
- D is a linear dimension
- μ is dynamic viscosity (Pa. s)
- ν is kinematic viscosity (m²/s); $\nu = \mu / \rho$

Considering the usual cut-in speed at 3-4m/s and cut-out at 25m/s for most wind turbines and by using the information obtained from Design of Support Structures for Offshore Wind Turbines, written by the civil engineer Jan Van Der Temper (2006), in which windspeeds according to the Beaufort scale are displayed as seen in Figure 4, the Reynolds number for the 0.66m chord line airfoil shown in Figure 3 is estimated.

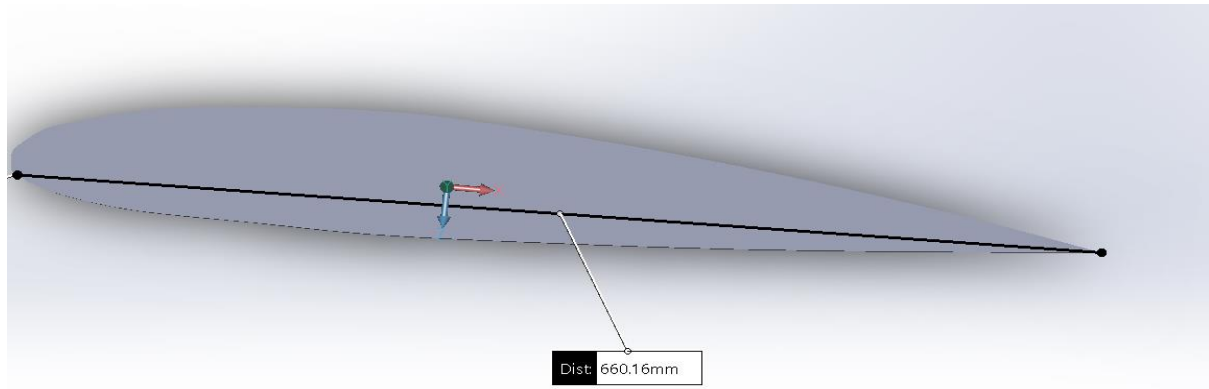


Figure 3. Chord length of the blade (Lake. Kelvin., 2020)

Beaufort	Knots	m/s	WMO description
0	< 1	< 0.2	Calm
1	1-3	0.3-1.5	Light air
2	4-6	1.6-3.3	Light breeze
3	7-10	3.4-5.4	Gentle breeze
4	11-16	5.5-7.9	Moderate breeze
5	17-21	8.0-10.7	Fresh breeze
6	22-27	10.8-13.8	Strong breeze
7	28-33	13.9-17.1	Near gale
8	34-40	17.2-20.7	Gale
9	41-47	20.8-24.4	Strong gale
10	48-55	24.5-28.4	Storm
11	56-63	28.5-32.6	Violent storm
12	> 64	>32.7	Hurricane

Figure 4. Wind speeds for an offshore wind turbine accordingly to the Beaufort scale (Van Der Tempel, J., 2006, p.31)

The viscosity, 0.00001789 Kg/m/s, as well as air density of 1.226 kg/m³, which represents standard parameters at sea level (15 °C) were also used (Munson, B.R, et al. 2013).

The Reynolds number was estimated for gentle breeze in which the velocities go from 3.4 to 5.4 m/s:

$$\bullet \text{ Reynolds number} = \frac{\rho V D}{\mu} = \frac{1,226 \times 5.4 \times 0.66}{0.00001789} = 246583.74$$

The obtained results show a high turbulent flow.

7. BOUNDARY LAYERS FLOW

Boundary layers flow around an airfoil is defined as the molecules of a fluid that are in direct contact with the surface of a solid while the fluid is moving past the object. Viscosity and the no-slip condition of the body's surface cause the formation of a layer (boundary layer) of retarded flow. The thickness of this layer is influenced by the Reynolds number, the higher this parameter is the thinner is the boundary layer. At the trailing edge, the boundary layer stops the contact with the surface, creating a wake flow past the body (Riveros, H.G. and Riveros-Rosas, D., 2010)

There exist some important factors to consider about boundary layers flow (Pijush K. Kundu, et al. 2012):

- Boundary flow can be classified in either laminar or turbulent flow. Compared to the laminar boundary layer, the turbulence boundary layer produces more shear stress. Laminar fluid flow occurs when a fluid's particles move along streamline in non-disrupted parallel layers. This type of flow is also called viscous flow. On the other hand, turbulent fluid flow occurs when

the particles do not travel in layers but mixing across the streamline causing chaotic property changes as well as high energy losses. To define the type of flow, the parameter Reynolds number is used, which must be less than 2000 to consider the flow to be laminar, between 2000 and 4000 is critical flow and greater than 4000 the flow would be turbulent.

- The pressure distribution across the surface and at the end of the boundary edge are equal. Therefore, the pressure on the boundary layers is dictated by the outer flow.
- Depending on increasing or decreasing pressure, the boundary layer will act in a different way. When the pressure is increasing, the laminar flow changes to turbulence, process called transition.

Nevertheless, if the boundary layer leaves the surface before due to an increase of pressure in downstream direction or in the case of an extremely curved surface, separation happens (*Riveros, H.G. and Riveros-Rosas, D., 2010*). Besides the fact that as well as laminar flows, turbulence flows also tends to separate, this last one resists the positive pressure gradient longer than laminar flow. This occurs because more high-speed fluid near the exterior face of the body is developed by the velocity profile in a turbulent boundary layer. In contrast, laminar boundary layer separates at very small pressure increase.

8. ANGLES FORMED BETWEEN THE BLADE AND WIND FORCES

To understand how a wind turbine generates electricity, it is important to mention the momentum theory, which is a theory applied to the actuator disk of a propeller to predict lift forces and power performance. If the propeller is rotating with an angular velocity, the blade will then have a velocity relative to the flow as well as a tangential velocity (*Rawson, K.J. and Tupper, E.C., 2001*).

Due to the inclination of the blade with respect to relative wind, various angles are formed. These angles can be appreciated in [Figure 5](#).

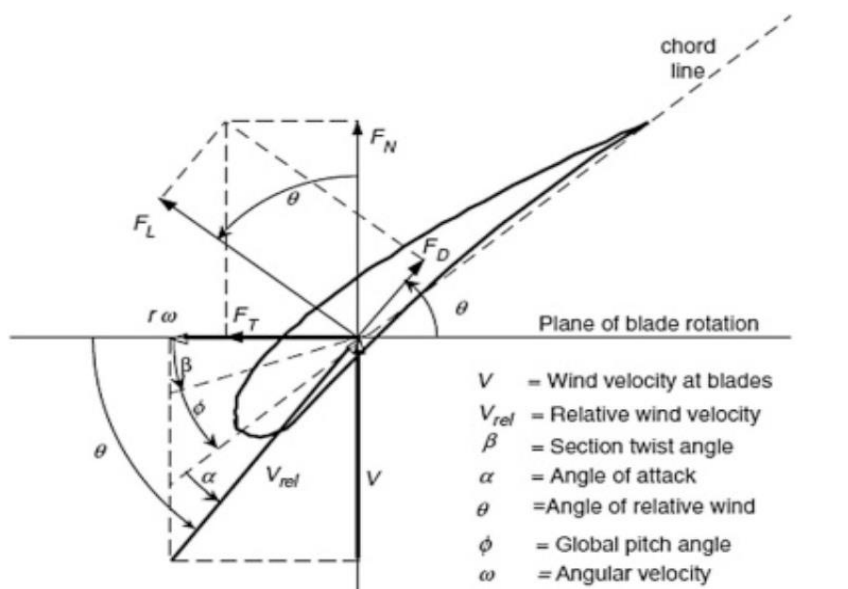


Figure 5. Forces acting on an airfoil section (*Jack Kelvin Ly, 2017*)

8.1 FLOW ANGLE

According to [Figure 5](#), the flow angle (θ) is the angle formed between the relative wind velocity vector (V_{rel}) and the plane of blade rotation.

To demonstrate how the flow angle along a wind turbine blade would change along the length, an example in which a windspeed of 15 m/s, a blade length of 10 meters and a rotational speed of one revolution per second has been used. To calculate the change in flow angle along the airfoil, two fundamental elements must be considered: initial linear free stream air velocity and rotational flow direction which is perpendicular to the wind due to the rotation of the blades. Hence, these two vectors form a triangle to which the trigonometric formula of hypotenuse can be applied to determine the angle.

$$\theta = \arctan \left(2\pi \times \frac{n \times 2r}{v} \right) = \arctan \left(2\pi \times \frac{1 \times 10}{15} \right) = 76.57 \text{ degrees}$$

where

- n is the rotational speed in rps.
- v is the wind speed in m/s.
- r is the radius in meters.

8.2 ANGLE OF ATTACK

The angle of attack calculates how effectively lift force is produced from the angle formed by the chord length of the airfoil with respect to the incoming wind. In [Figure 5](#), the angle of attack is represented by α . In contrast to a symmetrically shaped airfoil, in an airfoil with camber, lift will be produced even at 0 angle of attack. When the angle of attack increases, the lift force also increases up to a certain angle. However, when a separation of the flow happens due to surface friction because of viscosity at higher angles of attack and a drastically lift force decrease, the blade stalls. This condition is also related to the lift/drag ratio, as the drag force, which is the force parallel to the wind flow, increases drastically when the angle of attack reaches its maximum. In addition, there is also centrifugal force acting along the blade due to the rotation (*Jack Kelvin Ly, 2017*).

8.3 PITCH ANGLE: TWIST ANGLE AND GLOBAL PITCH ANGLE

The pitch angle is formed by two parts: twist angle and global pitch angle.

As Schubel, P.J. and Crossley, R.J. With noted in *Wind Turbine Blade Design (2012)*, with respect to the plane of rotation, the chord line of the cross section is given to a twisted called the section twisted angle ' β '. Moreover, to decrease blade manufacturing costs, the angle of twist might be reduced. However, this action could result in losses in the wind turbine's performance due to operations at angles of attack that are not ideal, in which lift to drag ratio is reduced (*Schubel, P.J. and Crossley, R.J., 2012*). On the other hand, to prevent the blade from reaching a stall angle, an additional angle named global pitch angle (ϕ) is given, which would modify the twist angle of every cross section (*Sudhamshu, A.R, et al. 2016*).

To show how the pitch angle of a blade must change along the length so that the blade does not stall, the graph in figure [Figure 6](#), which shows the Coefficient of lift v Angle of Attack, has been used:

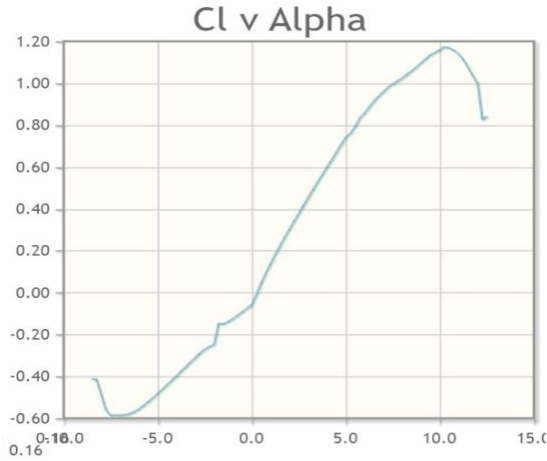


Figure 6. Coefficient of lift v Angle of Attack (Lake. K., 2020)

Two different lengths were used to calculate how the pitch angle would change to keep lift distribution reasonable constant while respecting stall considerations. Since the root end operates closer to stall than the tips, the pitch angle was calculated at two different lengths: 2 metres and 10 metres. Besides the fact that the flow angle has already been calculated at 10 metres, which was 76.57 degrees, calculations at 2 metres were also estimated:

$$\theta = \arctan \left(2\pi \times \frac{n \times 2r}{v} \right) = \arctan \left(2\pi \times \frac{1 \times 2}{15} \right) = 39.95 \text{ degrees}$$

Therefore, by having the flow angles at both lengths, the pitch angle can also be determined by subtracting the selected angles of attack at the preferred lift coefficient to the flow angles. The preferred angle of attack was 7.5 degrees at 1 coefficient of lift, as this is the most optimal angle of attack value before the blade stalls.

➤ At 10 metres:

$$\theta = \phi + \alpha$$

$$\phi = \theta - \alpha$$

$$\phi = 76.57 - 7.5 = 69.1 \text{ degrees}$$

➤ At 2 metres:

$$\theta = \phi + \alpha$$

$$\phi = \theta - \alpha$$

$$\phi = 39.95 - 7.5 = 32.45 \text{ degrees}$$

While the blade's pitch angle by the tip would need to be 69.1 degrees to keep a constant lift distribution, a 32.45 pitch angle closed to the rotor would be satisfactory to avoid stall.

9. CONCLUSION

Not only wind turbine tunnels and CFD are of vital importance now but also in the future infrastructures design optimisation. Additionally, neither of them should replace each other, what is more, they should combine with each other to counter the challenges presented by extreme natural events.

Moreover, based on the solutions obtained throughout the document, the following conclusions have been made:

- The high Reynolds number for a wind turbine of 0.66m chord length and a wind velocity of 5.4 m/s, reveals that the flow is turbulent. Furthermore, it also indicates small viscous forces, meaning that there would not be drag force.
- For a relative wind speed of 15 m/s, the adequate pitch angle should be 32.45 degrees at 2 meters from the base of the blade and 69.1 degrees at 10 metres to avoid reaching a stall angle. At other angles, the generated power would decrease.
- At angles of attack greater than 10 degrees the blade would eventually shut off.

- Païdoussis, Michael P., Stuart J. Price, and Emmanuel De Langre, 2010, *Fluid-structure interactions: cross-flow-induced instabilities*.
- McDonough, J.M., 2009. Lectures in elementary fluid dynamics: physics, mathematics and applications.
- Chen, T.Y. and Liou, L.R., 2011. Blockage corrections in wind tunnel tests of small horizontal-axis wind turbines. *Experimental Thermal and Fluid Science*, 35(3), pp.565-569.
- Wendt, J.F. ed., 2008. *Computational fluid dynamics: an introduction*. Springer Science & Business Media.
- El Mouhsine, S., Oukassou, K., Ichenial, M.M., Kharbouch, B. and Hajraoui, A., 2018. Aerodynamics and structural analysis of wind turbine blade. *Procedia Manufacturing*, 22, pp.747-756.
- Pires, O., Munduate, X., Ceyhan, O., Jacobs, M. and Snel, H., 2016, September. Analysis of high Reynolds numbers effects on a wind turbine airfoil using 2D wind tunnel test data. In *J. Phys.: Conf. Ser* (Vol. 753, p. 022047).
- Menon, E.S., 2015. Fluid flow in pipes. *Transmission Pipeline Calculations and Simulations Manual*, pp.149-234.
- Miller, M.A.M., 2018. *High Reynolds Number Horizontal and Vertical Axis Wind Turbine Experiments* (Doctoral dissertation, Princeton University).
- Rehm, B., Haghshenas, A., Paknejad, A.S. and Schubert, J., 2008. Situational problems in MPD. In *Managed Pressure Drilling* (pp. 39-80). Gulf Publishing Company.
- Van Der Tempel, J., 2006. Design of support structures for offshore wind turbines.
- Munson, B.R., Okiishi, T.H., Huebsch, W.W. and Rothmayer, A.P., 2013. *Fluid mechanics* (p. 147). Singapore: Wiley.
- Riveros, H.G. and Riveros-Rosas, D., 2010. Laminar and turbulent flow in water. *Physics Education*, 45(3), p.288.
- Pijush K. Kundu, Ira M. Cohen and David R. Dowling. *Mechanics*, F., 2012.
- Rawson, K.J. and Tupper, E.C., 2001. *Basic Ship (Fifth Edition)*, Butterworth-Heinemann

- Schubel, P.J. and Crossley, R.J., 2012. Wind turbine blade design. *Energies*, 5(9), pp.3425-3449.
- Sudhamshu, A.R., Pandey, M.C., Sunil, N., Satish, N.S., Mugundhan, V. and Velamati, R.K., 2016. Numerical study of effect of pitch angle on performance characteristics of a HAWT. *Engineering Science and Technology, an International Journal*, 19(1), pp.632-641.
- Ly, Jack Kevin, "ANGLE OF ATTACK DETERMINATION USING INERTIAL NAVIGATION SYSTEM DATA FROM FLIGHT TESTS. " Master's Thesis, University of Tennessee, 2017. https://trace.tennessee.edu/utk_gradthes/4757
- Lake, K., 2020. Designing a wind turbine blade: Aerodynamic design
- Lake, K., 2020. Designing a wind turbine blade: Structural design