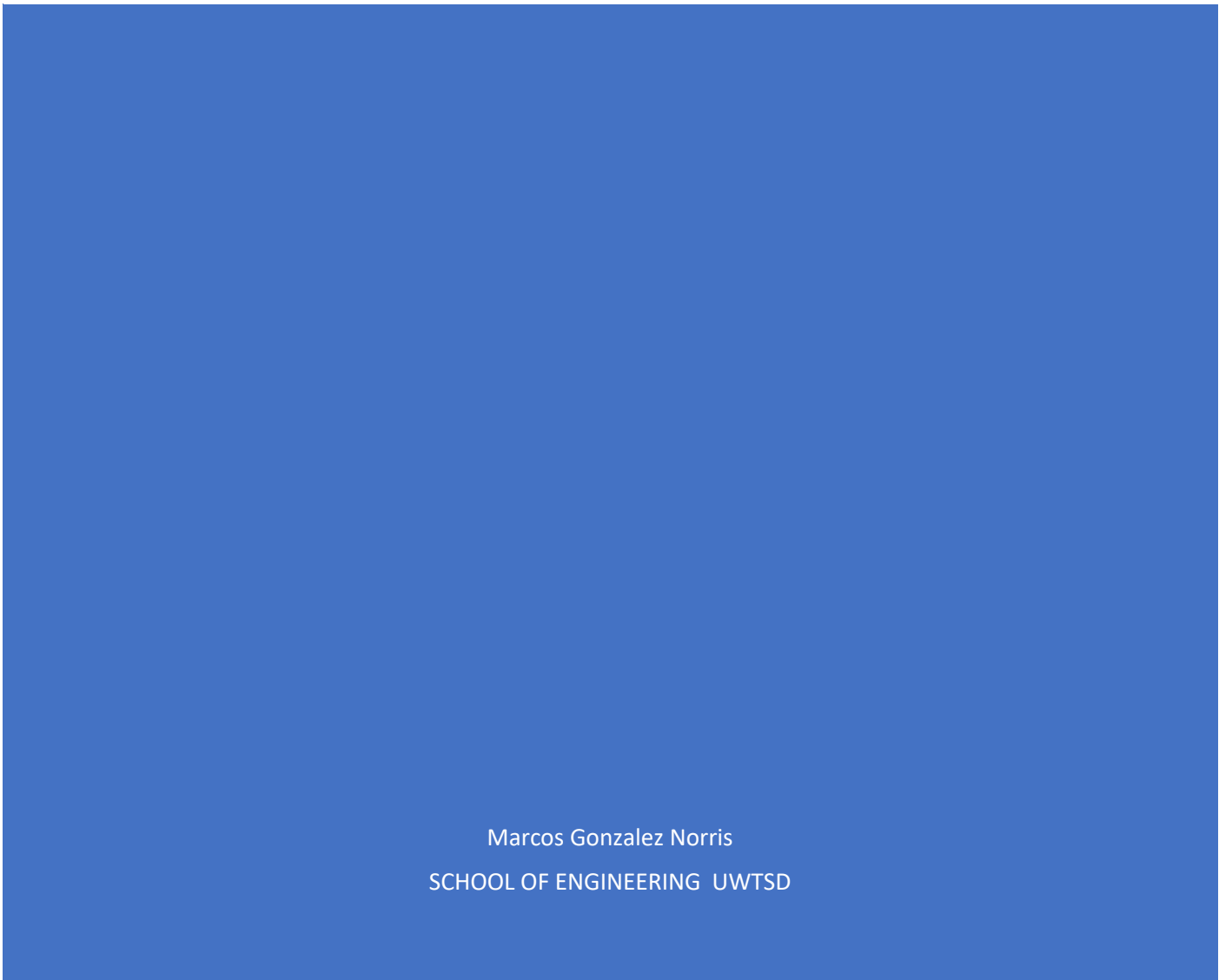




DESIGNING A WIND TURBINE BLADE: AERODYNAMIC DESIGN



Marcos Gonzalez Norris
SCHOOL OF ENGINEERING UWTSD

Contents

Introduction	1
Reynolds number	1
Calculating the Reynolds number for a wind turbine blade	2
Boundary flow	2
Flow angle for a wind turbine blade	3
Angle of attack and pitch	4
Conclusion	5
References	6
Equation 1: Reynolds number (Anon., 2020)	2
Equation 2: Reynolds number from set values	2
Equation 3: Angle of attack equation using pitch (Anon., 2020)	4
Figure 1: Figure showing resultant velocities caused by wind and tangential velocities	3
Table 1: Table showing flow angle at different lengths	4
Table 2: Table showing pitch angles at different lengths	5

Introduction

A wind turbine works by using its aerodynamic shape to manipulate fluid flow so that it can convert kinetic energy into electrical energy. In order to design an efficient turbine, there must be an understanding on how the fluid flow is affected by the turbines shape and therefore how the turbine can be optimised so that the aerodynamic shape can manipulate the fluid flow efficiently. This report investigates how the Reynolds number and the boundary layers effect fluid flow around the wind turbine blade and has a result how this affects the aerodynamic design of the blade with regards to angle of attack and pitch. This report looks into the Reynolds number, boundary flow and the effects these have on a wind turbine blades aerodynamic design in order to achieve maximum energy transfer from the fluid flow to the generation of lift at the blades.

Reynolds number

Investigations into a dimensionless value for the prediction into fluid flow began with the work of Sir George Stokes (1819-1903). Stokes' work on viscous fluid flow had led to the stokes flow, named after Sir George, himself, it could predict fluid flow when the Reynolds number was so low it could be presumed to be 0. It was in 1883 when Osbourn Reynolds, an Irish scientist, discovered what is now known as the Reynolds number (Anon., 2020).

The Reynolds number is a dimensionless value that refers to the ratio of inertial forces and viscous forces. It is used to determine what kind of flow a fluid will experience when passing across the

surface of an object. If the Reynolds number is lower than 2300 then the fluid flow is considered to be laminar. A laminar flow is a fluid flow which is smooth along a surface, it has multiple layers of fluid flow known as lamina, hence the name laminar. It is also known as streamlined and is the preferred form of flow when trying to minimise skin friction drag due to its low energy state and uniform direction (Nave, 2005). If the Reynolds number is greater than 4000 then the fluid flow is known as turbulent. This is where the fluid does not flow in layers but travels between them due to turbulent fluid rising and turning in on itself (Engineering Toolbox, 2004). This turbulent flow can be good or bad depending on what the desired outcome is. Its high energy state can be used to delay certain aspect of drag and therefore aid in the production of lift. However, this turbulent state can also create more drag depending in the scenario.

The Reynolds number can be determined by using the following equation:

$$Re = \frac{\rho VL}{\mu}$$

Equation 1: Reynolds number (Anon., 2020)

Where ρ is the density of the fluid, V is the velocity of the fluid, L is the significant length and μ is the dynamic viscosity.

The Reynolds number has a very important impact on the lift to drag ratio within a wind turbine blade. For the blade in a wind turbine to rotate, the lift created by the fluid running across the blade must be greater than the drag the fluid produces. In general, a larger Reynolds number means there is a larger lift to drag coefficient, thus, making it more efficient.

Calculating the Reynolds number for a wind turbine blade

By measuring the solidworks model of the wind turbine blade we can see that the chord length of the blade is 665mm or 0.665m. As this is the length that the fluid flows across it will be the value used within the calculations. The density of the fluid will be 1.2929 kg/m³ which is the density of dry air at sea level (Elert, 2006). The average wind speed in an offshore wind farm is around 15-24knots which is between 7.7 m/s and 12 m/s (Wong, et al., 2010). To make calculations simpler, 10 m/s has been used as the fluid velocity. For a temperature of 2 degrees Celsius the dynamic viscosity of air is 1.725 x 10⁻⁵ kg/m s and for a temperature of 27 degrees Celsius air has a dynamic viscosity of 1.846 X 10⁻⁵ kg/m s (toolbox, 2005). As temperatures will vary between these two points a dynamic viscosity of 1.8 x 10⁻⁵ kg/ m s has been used to calculate the Reynolds number.

If we take all of these values and apply them to equation 1, we get the following equation:

$$Re = \frac{1.2929 \times 10 \times 0.665}{1.8 \times 10^{-5}}$$

Equation 2: Reynolds number from set values

Equation 2 gives us an answer of 477654.7 this would show significant turbulent flow.

Boundary flow

When a fluid flows over a surface the particles closest to the surface stick to the material. The layer of particles flowing above collide with these stationary particles and therefore have a reduction in velocity. With each layer of particles, the reduction in velocity decreases until there is no reduction at all. The area of fluid flow that has no reduction in velocity is known as the free stream. The area of fluid flow that has reduced fluid velocity is known as the boundary layer. The boundary layer acts like the surface of the object, making the free stream fluid flow around it and interact with it. As a

result of this, the shape and size of the boundary layer has a huge effect on how fluid interacts with an object. (Hall, 2015).

There are two main factors that create resistance or drag at the boundary layer, they are skin friction drag and pressure drag. The sum of these two resistances account for the overall drag of an object when subjected to fluid flow. The velocity changes within the boundary layer create internal shear stresses which behave similarly to how friction would hence the name “skin friction drag”. The greater these changes in boundary layer velocities the greater the shear stresses are and therefore the greater the drag. As a result of the fact that turbulent flow has greater velocity changes compared to laminar flow, it produces greater skin friction drag than a laminar flow.

Pressure drag is a resistance that forms due to something known as boundary layer separation. Boundary layer separation is where the fluid in the boundary layer reduces in velocity, causing an increase in pressure and thus the boundary layer begins to separate from the surface of the object. As the fluid slows down it can also begin to travel in the opposite direction to the fluid flow causing resistance i.e., drag. In order to reduce the effects of pressure drag you want to delay boundary layer separation as much as possible. In laminar flow the change of velocity between the surface to the free stream is much slower meaning that the boundary layer separation can occur earlier compared to that of a turbulent flow (Rainer, 2016).

Due to these two types of drag it is clear to see that there are benefits and draw backs to both laminar and turbulent flow. However, at lower velocities where the velocity differences are not that great and therefore the internal shear stresses are not that great, turbulent flow is probably the most desired state in order to push back the separation point. This in turn should create the least amount of drag and therefore have a higher lift to drag ratio.

When looking at wind turbine blade this boundary layer separation is key to having an efficient blade. If the boundary layer separates too early along the blade, then it will not produce the optimum amount of lift which in turn means that there are energy losses occurring due to drag. If you are to design an aerodynamically efficient blade, then this must be considered so that the design can avoid this boundary separation and therefore avoid excess drag.

Flow angle for a wind turbine blade

The flow angle is the angle at which the fluid interacts with the blade with reference to the wind velocity and the rotational velocity. The following figure shows what this would look like.

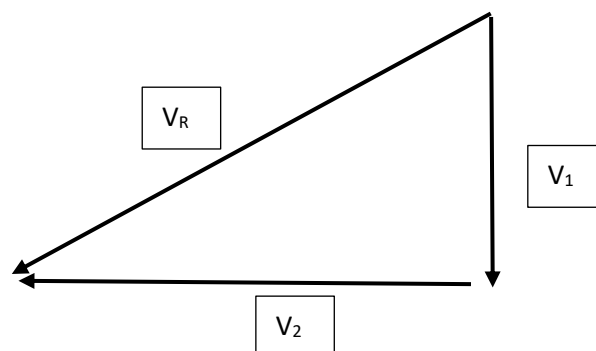


Figure 1: Figure showing resultant velocities caused by wind and tangential velocities

Where V_1 is the velocity of the wind and V_2 is the tangential velocity at the very apex of the blades trajectory so that it is perpendicular to the wind. V_R is then the resultant velocity of these two motions, meaning that the angle between V_1 And V_R would be the flow angle. In order to find this angle, we can use simple trigonometry and basic mathematics. By using the inverse of tan and dividing V_1 by V_2 we can find the angle between V_R and V_2 . As all angles in a triangle add up to 180 degrees and we know that one of the angles is 90 degrees as V_2 is perpendicular to V_1 then we take the angle calculated away from 90 in order to get the flow angle. (Anon., 2020)

However, the tangential velocity changed as you move along the blade due to the fact that the larger the radius of rotation, the greater the distance is needed to travel in the same amount of time. As a result of this the table below shows what the tangential velocities are at each meter of blade length and therefore what the flow angles are as you move along the blade. The tangential velocities are worked out by calculating the circumference of the rotation and then dividing it by the time taken for one full rotation. The results from the table below are taken with a wind speed of 15m/s, a rotational speed of 1 rotation per second and a blade length of 10m.

Blade length (m)	Tangential velocity (m/s)	Flow angle (°)
1	6.283185307	22.72778733
2	12.56637061	39.95486252
3	18.84955592	51.48811275
4	25.13274123	59.16993619
5	31.41592654	64.47716565
6	37.69911184	68.30301603
7	43.98229715	71.16824696
8	50.26548246	73.38408022
9	56.54866776	75.14394872
10	62.83185307	76.57295824

Table 1: Table showing flow angle at different lengths

As we can see from table 1 the flow angle increases as the blade gets longer. This is because the longer the blade is the higher the velocity and therefore the greater its influence on the flow angle. This means that the way the air flows around the blade changes drastically as you move along the blade and therefore the coefficient of lift would also change.

Angle of attack and pitch

By using data from the assignment brief we can see that a maximum lift coefficient is reached when the angle of attack is around 10 degrees before it begins to stall. In order for the wind turbine blade to be aerodynamically efficient the blade must keep this angle of attack throughout its length. However, as the flow angle changes along its length the angle of attack also changes. Therefore, the pitch of the blade must also change along the blade. The following equation shows the relationship between angle of attack and pitch.

$$\alpha = \theta - \tan^{-1}\left(\frac{V_1}{V_2}\right)$$

Equation 3: Angle of attack equation using pitch (Anon., 2020)

Where α is the angle of attack, θ is the pitch, V_1 is the wind velocity and V_2 is the tangential velocity.

By rearranging equation 3 to make the pitch the subject and by defining the angle of attack as 10 degrees we can then calculate the pitch of the blade at any given length. The table below shows this for every meter of the blade using the same conditions that were used in table 1.

Blade length (m)	Pitch (°)
1	77.27221
2	60.04514
3	48.51189
4	40.83006
5	35.52283
6	31.69698
7	28.83175
8	26.61592
9	24.85605
10	23.42704

Table 2: Table showing pitch angles at different lengths

The angle of attack is the angle at which the fluid hits the blade relative to the blade itself. As shown by table 1, flow angle is constantly changing throughout the length of the blade, meaning that if nothing else changes the angle of attack would also change. As a result of this the pitch needs to change to keep the same angle of attack to avoid stalling.

We can see from table 2 that the pitch decreases as the blade gets longer. Mathematically this relationship can be seen with equation 3 as the blade length has a direct correlation with the resultant velocities.

The reason the angle of attack has an effect on whether or not the blade stalls is down to what was previously discussed with regards to the boundary layer separation. By increasing the angle of attack the blade sees an increase in pressure differences and thus an increase in lift. However, if the pressure difference is too great then the fluid flow slows down too much, and boundary flow separation occurs earlier. This in turn changes the overall shape of the blade causing a reduction in lift and therefore the wing stalls.

Conclusion

In conclusion, it is clear to see that the Reynolds number has a very big role in determining how the boundary layer around an aerofoil behaves. This boundary layer in turn, has a huge effect on whether the aerofoil stalls, creates lift or is efficient enough.

We can see from simple calculations that the Reynolds number in a wind turbine is very high, thus proving that a wind turbine blades boundary flow is under turbulent flow.

When looking at wind turbines its important to look at these parameters to aid in the aerodynamic design in order to prevent unnecessary energy losses caused by excess drag or wing stalling. This can be done by calculating the flow angle at every point within a wind turbines blade length and then

calculating the require pitch to prevent stalling. From the results gathered we can see that overall, the pitch decreases, and flow angle increases as the blade gets longer.

References

Anon., 2020. *Blade Element Theory for Propellers*. [Online]

Available at: <http://www.aerodynamics4students.com/propulsion/blade-element-propeller-theory.php#:~:text=Blade%20Element%20Theory%20for%20Propellers,use%20of%20Blade%20Element%20Theory.&text=The%20resulting%20values%20of%20section,overall%20performance%20of%20the%20pro>

[Accessed 02 01 2021].

Anon., 2020. *simscale*. [Online]

Available at: <https://www.simscale.com/docs/simwiki/numerics-background/what-is-the-reynolds-number/>

[Accessed 27 december 2020].

Anon., . *Dynamic, Absolute and Kinematic Viscosity*. [Online]

Available at: http://www.engineeringtoolbox.com/dynamic-absolute-kinematic-viscosity-d_412.html

[Accessed 5 1 2021].

Elert, G., 2006. *Density of Air*. [Online]

Available at: <http://hypertextbook.com/facts/2000/RachelChu.shtml>

[Accessed 5 1 2021].

Hall, N., 2015. *Nasa*. [Online]

Available at: <https://www.grc.nasa.gov/www/k-12/airplane/boundlay.html>

[Accessed 05 01 2021].

Nave, R., 2005. *Laminar Flow*. [Online]

Available at: <http://hyperphysics.phy-astr.gsu.edu/hbase/pfric.html>

[Accessed 5 1 2021].

Rainer, 2016. *Boundary Layer Separation and Pressure Drag*. [Online]

Available at: <https://aerospaceengineeringblog.com/boundary-layer-separation-and-pressure-drag/>

[Accessed 09 01 2021].

toolbox, E., 2005. *Dry air properties*. [Online]

Available at: http://www.engineeringtoolbox.com/dry-air-properties-d_973.html

[Accessed 5 1 2021].

Wong, K.-F. V., Hutley, T. & Salgado, E., 2010. *Offshore Wind Power and its Potential for Development in the West Wind Drift*. [Online]

Available at:

<https://proceedings.asmedigitalcollection.asme.org/proceeding.aspx?articleid=1616271>

[Accessed 5 1 2021].