



Prifysgol Cymru
Y Drindod Dewi Sant
University of Wales
Trinity Saint David

Designing a kiteboard: Fluid flow assessment - ACMA6010

Dan Ward Daniel Ward – 1504426

Word count – 2988 words

Contents

1. Introduction	2
2. What is Reynolds number?	2
3. Boundary flow around the board.....	4
Under the board.....	4
Bernoulli's principle.....	5
Tail of the board	6
4. Implications of Using a Carbon Reinforced Polymer Board	7
5. Modelling Techniques	7
6. Experimentation, testing and proving the model	9
Wind Tunnels	9
Open Return wind tunnels.....	10
Closed return wind tunnels.....	11
Why use a full -size model for testing?	11
7. Summery	11
8. References.....	13

1. Introduction

This paper will investigate the fluid flow around a kite surfing board, it will discuss why Reynolds number is important and estimate the Reynolds number.

It will then move onto discussing the flow around the boundary of the board, the board will be interacting with essentially two fluids. This will look at the separation and turbulence around the board.

The board will be constructed from Carbon Reinforced polymers, which will have some implications on the board's performance. This material will be interrogated to determine whether it is best suited to this application and evaluate past present and future developments.

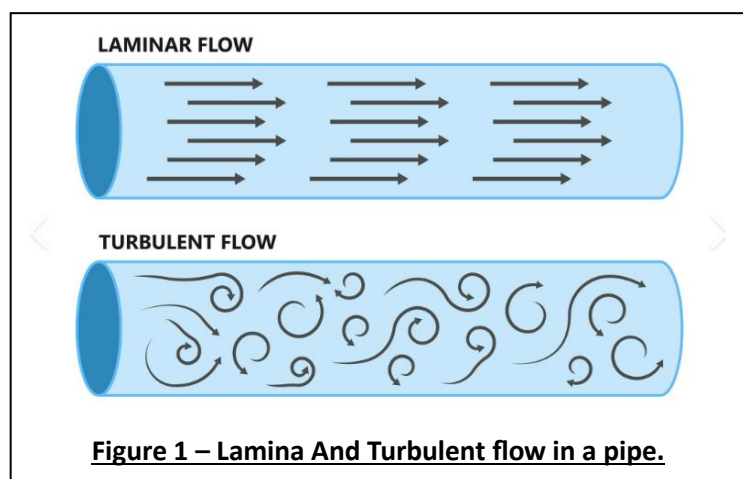
It will then move onto the modelling techniques used to model the board pre-production. This will give opportunity to discuss the advantages and disadvantages of 3dmodelling and what processes are used to evaluate the validity of the results as well as evaluate the forces acting up on the board.

A prototype of the board will be made and tested in a laboratory, this paper will determine how the experiments will be constructed and what is expected of the results.

2. What is Reynolds number?

Named after Osbourne Reynolds of the late 1800's, represented in engineering and physics as Re , Reynolds Number represents the ratio between the inertia forces and viscous forces which act within fluids that are subject to flow or internal movement of the fluids. Simply meaning that

When Re is less than 2000 it is considered low, the flow is generally laminar when passing over a flat or aerodynamic surfaces. This means that the flow has layers that are parallel from one another each layer moves smoothly past the adjacent layers



with little to no mixing between each layer as demonstrated in figure 1. (Flow : applications of fluid mechanics, 2021)

over 2000 and the fluid becomes turbulent (Serdar Genc, 2012).

To calculate Reynolds number the following equation is used:

$$Re = \frac{\rho \cdot u \cdot L}{\mu}$$

When designing a kite board the following assumptions are made:

- Average density(ρ) of ocean water is 1027kg/m³
- The fluid velocity(u), which will be the amount of water passing under the board per second, calculated as the volume of water passing under the board per second. A typical kiteboard has a surface area of 1.25m x 0.4m therefore the total board area is 0.5m² then we will assume that approximately 75% of the board will be under water at an angle of 15° at the surface, using Pythagoras theorem we can calculate that the tail of the board will be 0.243m below the water surface. Times this by the width of the board to calculate the area of water effected by the board passing over (0.0338m²), times by the speed of the board which is travelling (17.9m/s) give the volume of water per second which in this case is 0.694m³/s.
- The third thing to be found is the characteristic linear dimension which was already calculated to determine the flow velocity, this the distance in front of the object in which a fluid flows or average cross section of the area in which a fluid flows though in our case (0.097m).
- The determine the Dynamic Viscosity of sea water, this varies depending of the temperature of the fluid and the amount of sault in the water. For this instance, we will use the average temperature of the world sea water which is 20° in warm areas of the world where most kiteboarding takes place and the average salinity level which is about 35g/kg. Using the MIT.edu web site table and some quick arithmetic we can say that the dynamic viscosity of sea water in the world is approx. 0.959g/kg (Kishor G. Nayar, 2016).

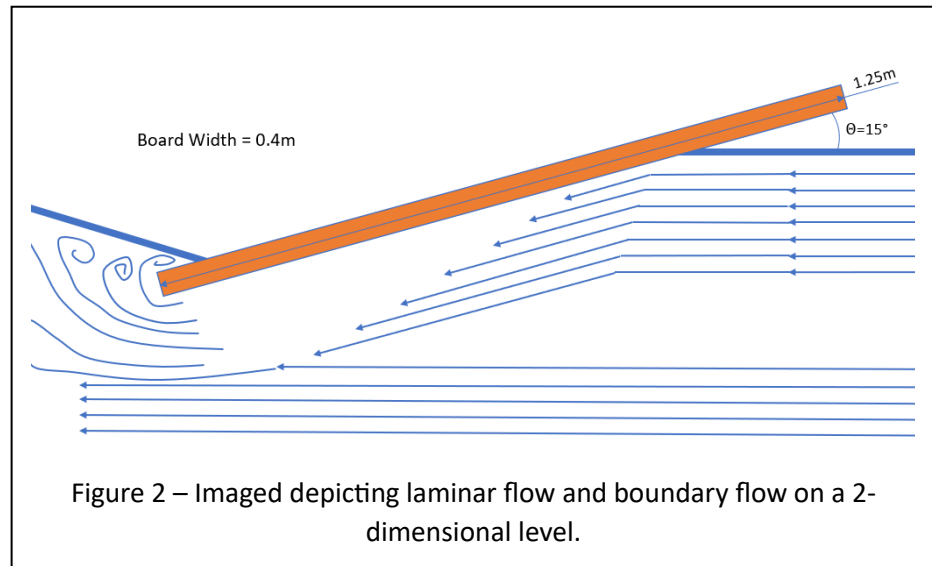
$$Re = \frac{1027 \times 0.694 \times 0.2}{0.959} = 53.545$$

In this case Re equals 53.545, which is considered a very low Re value therefore the flow is laminar.

3. Boundary flow around the board

Under the board

As discussed previously the low Reynold's number indicates a laminar flow under the surface of the board, meaning that the flow is essentially a



sequence of layers of molecules flowing parallel to the board (Munson, 2013). The layers of fluid furthest away from the board are traveling faster than those which are closest to the board this is depicted by the shorter arrows close to the board and longer blue arrows in Figure 2.

This is caused by the molecules closest to the board sticking to the board's surface, this in turn causes the next layer to collide with these molecules and slow them down. Each layer is then slowed down less and less until the flow of the fluid is at its maximum or in this its resting state, as shown in Figure 3.

This however only explains what is happening directly under the board, due to the displacement of water there is a slight increase in pressure. Meaning that at a point somewhere directly between the surface of the board moving over the water and the water which is not moving at all. The fluid is moving ever so slightly faster than the board itself. And we can calculate this using Bernoulli's equation.

Bernoulli's principle

Bernoulli's Principle is one of the fundamental principles of fluid dynamics. When using Bernoulli's principle one is making the following assumptions. The flow of the fluid is steady state, frictionless and streamline, its density is constant and there is no change in temperature, heat transfer or change in work (Bhattacharyya, 2024).

Bernoulli's principle is shown below equation and it represents the pressure plus the velocity plus gravity:

$$P_1 + \frac{1}{2}\rho v_1^2 + \rho gh_1 = \text{constant}$$

Where:

$P = \text{Pressure}$

$\rho = \text{Density}$

$v = \text{Velocity}$

$h = \text{height}$

The equations show that the total energy in a system remains constant therefore if the fluid flow increases, the pressure decreases.

For this instance, there is no difference in potential energy therefore that can be ignored, this leaves pressure and velocity. As the velocity increases the pressure under the board decreased creating lift and helping move the board /closer to the surface of the water in the same way as an aeroplane moves through the sky.

Bernoulli's Principle does have 2 prominent limitations, the first of which is that it can only be applied when discussing steady state flow where the speed and direction of the fluid remain a constant over time, this is due to the fact that unsteady flows are much more complex as the pressure distribution and velocity can change faster.

The second is that it assumes that the fluid material is inviscid. This means that the material has no internal friction which leads to small eddies and a level of turbulent flow which is witnessed in real-life scenarios.

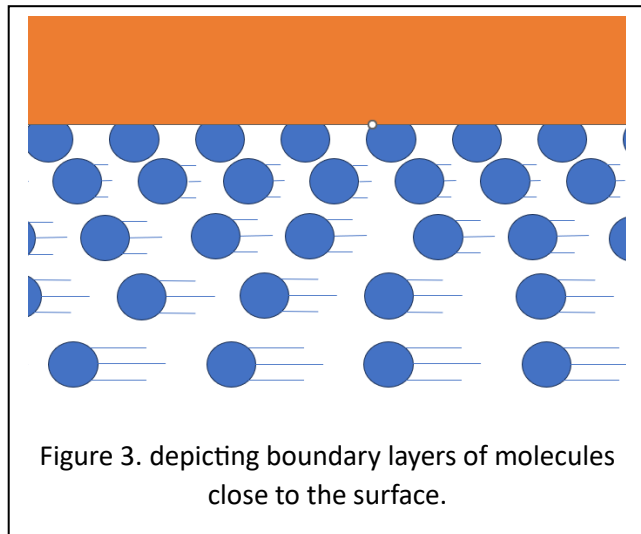


Figure 3. depicting boundary layers of molecules close to the surface.

Regardless of these limitations Bernoulli's Principle is a fundamental principle of fluid dynamics and remains essential for understanding how the fluid flows around the underside of the board.

Tail of the board

When looking closer to the flow at the rear or tail of the board, the flow of the fluid becomes turbulent. This is caused by separation of the laminar flow which in turn causes turbulent flow behind the board. Coupled with the friction within the fluid/colliding of the molecules a pressure decrease is witnessed as the fluid passes the very tail of the board.

Looking at Figure 4, point A, the molecules touching the board are at 0 velocity in comparison to the board, as the fluid goes over the rear tip of the board (Point B) more and more molecules stick to the board and the point at which the molecules are at 0 velocity is further away from the board. As the fluid passes over the tip of the board the pressure drops, some of the molecules fall into negative velocity

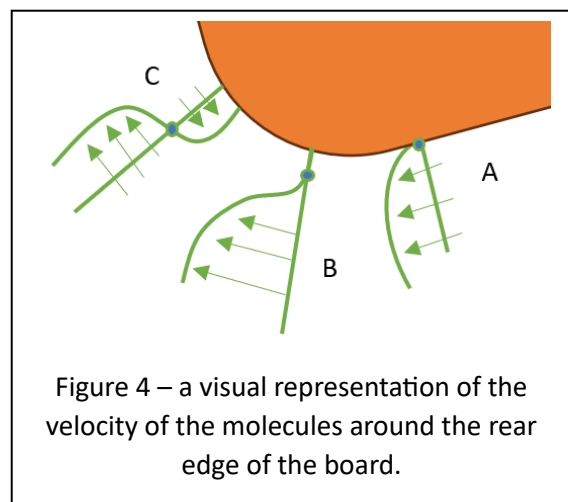


Figure 4 – a visual representation of the velocity of the molecules around the rear edge of the board.

and begin to get drawn back into the rear edge of the board creating small vortexes or turbulent fluid flow (Point C).

In short, this negative flow/turbulence coupled with energy losses through the friction within the fluid causes drag a negative pressure which pulls the board backwards.

4. Implications of Using a Carbon Reinforced Polymer Board

Using carbon fibre has a number of implications, some positive and others negative. Carbon fibre is expensive, difficult to manufacture at scale and is prone to delamination and cracking when impacted or when put under cyclic loading. These can be overcome by increasing the number of fibres/layers used in the material, laying the fibres in the direction in which the most strain/stress is applied on the board.

However, it is also light weight, can be formed in almost any shape and has a high weight to strength ratio (Hearn, 1997).

According to many kiteboarding enthusiasts carbon fibre boards are preferred by professional kiteboarders due to their responsiveness, precession and “pop”. This is due to Carbon fibre reinforced polymers are generally very flexible, this allows riders to use the spring in the board to get more height when hitting waves, much like gymnasts use a spring board to jump over large objects.

Another very important factor to the using carbon fibre for this application is that one can alter the surface finish of the board (Hearn, 1997). This is important because you can manipulate or roughen the finish which can reduce the friction, creating turbulence along the underside of the board creating small eddies. These eddies act like little marbles when traveling at speed.

5. Modelling Techniques

There are a number of ways to model kitesurfing boards, traditionally wood and bamboo boards have been made by feel and in many ways as an educated guess.

More recently engineers would model the board using CAN software and run the model through a software package that uses Computational Fluid dynamics (CFD), which uses numerical algorithms to solve tens of thousands of equations to simulate the interaction of

fluids flow around an objects surface. Navia-Stokes equations are the basis of this type of software (Bhattacharyya, 2024) (Danchin, 1965).

Not anyone can use this software as the user must understand the fundamental concepts of fluid dynamics. Understanding the equations such as the Navi-Stokes equations and boundary conditions is essential, as well as having a good understanding of meshing.

The Navia-Stokes equations must be solved simultaneously at each node on a two-dimensional plane. The Navia-Stokes equations are below:

$$\begin{aligned}\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z} &= 0 \\ \rho \left[\frac{\partial u}{\partial t} + \frac{\partial u}{\partial x} u + \frac{\partial u}{\partial y} v + \frac{\partial u}{\partial z} w \right] &= -\frac{\partial p}{\partial x} + \mu \left(\frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2} + \frac{\partial^2 u}{\partial z^2} \right) + \rho g_x \\ \rho \left[\frac{\partial v}{\partial t} + \frac{\partial v}{\partial x} u + \frac{\partial v}{\partial y} v + \frac{\partial v}{\partial z} w \right] &= -\frac{\partial p}{\partial y} + \mu \left(\frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2} + \frac{\partial^2 v}{\partial z^2} \right) + \rho g_y \\ \rho \left[\frac{\partial w}{\partial t} + \frac{\partial w}{\partial x} u + \frac{\partial w}{\partial y} v + \frac{\partial w}{\partial z} w \right] &= -\frac{\partial p}{\partial z} + \mu \left(\frac{\partial^2 w}{\partial x^2} + \frac{\partial^2 w}{\partial y^2} + \frac{\partial^2 w}{\partial z^2} \right) + \rho g_z\end{aligned}$$

First the user will need to take on some pre-processing, this is where the user cleans up the CAD model by removing any un-necessary features, simplify complex shapes, set mechanical properties of the fluid, such as density, viscosity temperature etc, use a mesh generator to create a mesh which is intricate enough to accurately simulate what is happening in real life but isn't too small that it uses too much processing power and takes a long to simulate. Engineering often use more intricate mesh sizes in areas that are more likely to affect the fluids most relevant to the application. In this case kiteboards usually have small fins on the underside of the board will have small features to keep the board travelling in the longitudinal direction, as we are able to simulate the flow in one direction and these features do not impact the fluid flow dramatically in this direction these are removed to reduce the amount of computational power thus reducing the time it takes to simulate.

The user would then choose the correct solver requirements. Again, requiring knowledge of fluid dynamics to answer questions such as is the flow steady state, transient, laminar or turbulent, and configuring the solver by specifying convergence criteria, time steps and

turbulence models if required. In this case the fluid will be steady state as the board will be traveling relatively slowly and the waves and direction of current will not have enough of an influence on the flow.

It is then time to run the solver, this is where the computer takes over. An engineer will still need to monitor the convergence and adjust settings as required. When complete the user can analyse the results, the programme will give a visual output of the velocity vectors, pressure contours etc. as well as give the user the ability to extract the relevant numerical data and validate against experimental data or theoretical predictions.

Small adjustment will be made to alter the flow to get the optimum performance of the board, the simulation will be re-run with the changes and this step will repeat until the board is at the optimum performance.

6. Experimentation, testing and proving the model

The designer or design team now have a representation of how the fluids act around the boundary layer of the board in the conditions specified. They would now aim to verify and validate what they have created on their simulation to show that it accurately represents what is happening in a real world scenario. The only way that this can be done is by experimentation and testing of the model.

To do this, engineers would need to build an accurate prototype of the board and testing it in a wind tunnel to test the hydrodynamic conditions of the board and prove the geometry of the board acts in the way predicted by the CFD simulation software.

Wind Tunnels

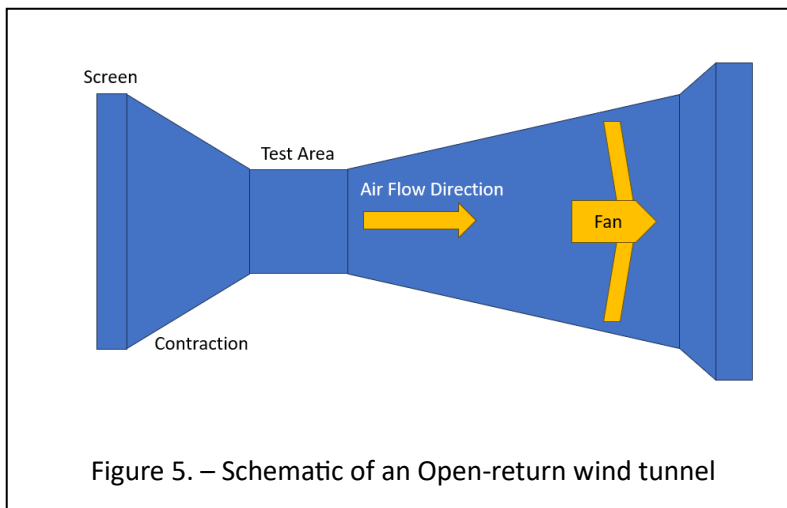
As stated in a magazine called Tunnel "Wind tunnels are precision laboratory instruments that create a controlled simulation of simplified real-world conditions". They allow engineers and scientists to evaluate the aerodynamics of an object or group of objects. But with all test equipment and simulation tools there are imperfections and some tolerance must be expected.

As the kiteboard in question is low value and will not require a vast amount of testing there is not a requirement to build a stand-alone wind tunnel therefore commercially available test facilities will be discussed.

There are two main types of wind tunnel, there is an open return wind tunnel and closed return wind tunnel. Each have their own pros and cons.

Open Return wind tunnels

Also known as Eiffel tunnels or NPL tunnels, open return wind tunnels work by drawing air from the room in which it is situated through a cone. This means that the air moves fastest through the test area at low pressure.



Open return wind tunnels where the beginning of wind tunnels and designed by the Wright brothers during the birth the birth of human flight (Chamorro, 2009).

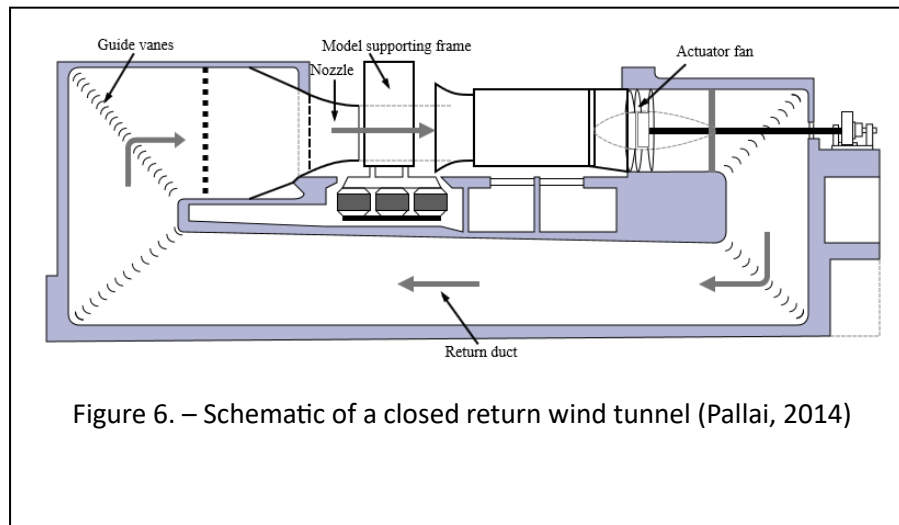
This type of wind tunnel is known to be noisy due to air flow quality in the test section

this is mitigated by using flow straighteners or screens, which forces the into a more laminar flow and ensuring that the wind tunnel is away from object in the room and the room is large enough so that the boundary layers created near the walls are not affecting air flow (Okamoto, 2011).

One advantage of the open loop system is that the test can be larger in relation to the entire system. The largest wind tunnel in terms of the test area is located in NASA Ames in California with a test area of 80x120ft (Dymott, 2022).

Closed return wind tunnels

A more efficient alternative to the open return wind tunnel due to its closed design the air flow inside a closed return wind tunnel is recycled therefore it uses less energy than a closed



return wind tunnel.

Closed return wind tunnels are also known for their good quality air flow inside the test area. This is due to the level of control that can be accomplished throughout the

system.

The main down side of using a close return wind tunnel is the cost, this is largely due to the vast amount of space required to house the wind tunnel.

Why use a full-size model for testing?

When verifying a model in a wind tunnel it is always best to use a full-scale model, the reason for this is because of how air flows around it. There are only three elements which can be changed to achieve similar results t these hard to achieve successfully which are, velocity, pressure and fluid density. To change the pressure or air density within a wind tunnel is costly as the entire wind tunnel would need to become a sealed unit.

Determining what type of wind tunnel is right for the application is large a matter of cost and location. Given that kite boards are relatively cheap in comparison to the automotive or aerospace industries who are the main users of them. Choosing a window tunnel near to the design location and weighing that up to a wind tunnel that is commercially available is most advantageous over building or travelling overseas.

7. Summery

Reynolds number was calculated to determine that the flow around the board was laminar Bernoulli's equation was used to calculate the amount of force acting on the board.

The flow around the boundary of the board, was determined to be lamina on the underside of the board and as the fluid past the rear of the board the pressure dropped separating the laminations forcing the fluid flow to become turbulent.

Using FRP's to the construction of the board allowed for almost any shape to be made but limited the scalability of production and increased the cost meaning this type of board would only be sold to enthusiasts and professional kiteboarders.

Modelling techniques used to model the board pre-production include CFD followed by the due of a wind tunnel. Different types of wind tunnel we discussed and due to cost it was determined that an open return wind tunnel was not required for this application and that no alterations to the test area was required due to cost implications.

8. References

- Barnard., R. (2009). *Road vehicle aerodynamic design : an introduction*. Alma: MechAero.
- Bhattacharyya, S. (2024). *Applications of computational fluid dynamics simulation and modeling*. London: IntechOpen. Retrieved from <https://doi.org/10.5772/intechopen.94665>
- Chamorro, L. a.-A. (2009). Wind-Tunnel Investigation of Wind-Turbine Wakes: Boundary-Layer Turbulence Effects. *Boundary-layer meteorology*, 129–149. Retrieved from <https://doi.org/10.1007/s10546-009-9380-8>
- Danchin, R. &. (1965). Compressible Navier–Stokes system: Large solutions and incompressible limit. *Advances in Mathematics*, 320, 904–925. Retrieved from <https://doi.org/10.1016/j.aim.2017.09.025>
- Douglas, J. F. (2011). *Fluid mechanics* (Sixth ed.). Harrow: Prentice Hall.
- Dymott, G. (2022, May 30). *NASA Full Size Wind Tunnels*. Retrieved from Airshaper.com: nasa-full-size-wind-tunnels
- Flow : applications of fluid mechanics*. (2021). Cambridge: Cambridge University Press.
- H. Versteeg, W. (2007). *An Introduction to Computational Fluid Dynamics: Finite Volume Method*. Harlow, England: Pearson.
- Hearn, E. J. (1997). *Mechanics of materials 2 : an introduction to the mechanics of elastic and plastic deformation of solids and structural materials*. Oxford: Butterworth-Heinemann.
- Kishor G. Nayar, M. H. (2016, December 21). *Seawater Thermophysical Properties Library*. Retrieved June 12, 2024, from <http://web.mit.edu/seawater>: https://web.mit.edu/seawater/2017_MIT_Seawater_Property_Tables_r2b_2023c.pdf#:~:text=Dynamic%20viscosity%20of%20seawater%20x%20103,P0%20%28~%201%20atm%20or%200.10325%20MPa%29
- Macchia, S. a. (2017). A simple wind tunnel to analyse Bernoulli's principle. *Physics education*, 51(1). Retrieved from <https://doi.org/10.1088/1361-6552/52/1/013004>
- Munson, B. e. (2013). *Fluid mechanics*. (Seventh ed.). Singapore: Wiley.
- Okamoto, S. (2011). *Wind tunnels*. (S. Okamoto, Ed.) Rijeka, Croatia: IntechOpen.
- Pallai, A. (2014, January 20). *Unit I Open and Closed Circuit Wind*. Retrieved from wtttaerojjcet.blogspot.com: <https://wtttaerojjcet.blogspot.com/2014/01/unit-i-open-and-closed-circuit-wind.html>
- Serdar Genc, M. (2012). *Low Reynolds Number : Aerodynamics and Transition*. Genç. Rijeka: IntechOpen.