

Joel Lang

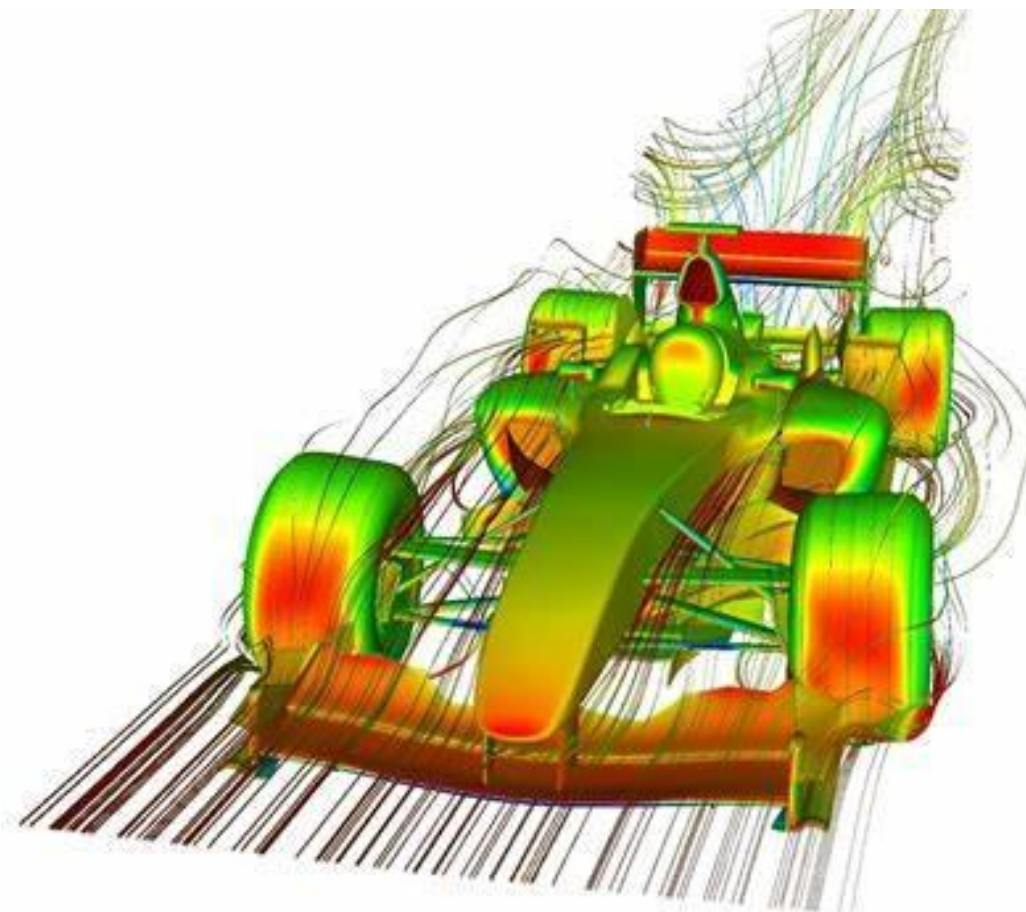
2014802

Structural & Fluid analysis

ACMA6010

Designing a tanker propeller: Fluid flow
assessment

Word count: 3308



Contents

Table of figures	3
1.	4
Reynolds number	4
Laminar vs turbulent flow	4
ρ =density.....	5
v =velocity	5
l =length	6
μ =viscosity.....	6
Other assumptions.....	6
Final Reynolds number	7
2.	7
Boundary flow	7
Design of propeller blade.....	8
3.	9
CFRPs.....	9
.....	9
CFRP propeller	9
Past developments.....	10
Current developments	11
Future developments.....	11
4.	11
CFD	11
Benefits of CFD.....	12
Limitations of CFD	12
Mitigating limitations of CFD	12
Alternative methods	13
5.	13
Validation of CFD	13
Water tunnels	14
.....	15
Data collection	15
Relation to CFD	15
Works Cited.....	16

Table of figures

Figure number	Description
1	Laminar vs. turbulent flow
2	Resultant Force and flow vectors of a propeller.
3	Chord length
4	boundary layer over airfoil
5	adverse pressure gradient
6	pressure regions and flow over airfoil
7	CFRP stress/strain vs. other materials
8	Hookes screw propeller
9	Bramah's screw propeller design
10	Propeller of the napoleon (1842)
11	CFD validation procedure
12	water tunnel schematic
13	validation experiment schematic

1.

Reynolds number

When considering fluid flow, it is essential to be able to predict whether the fluid flow will be turbulent flow or laminar flow. To do this Reynolds number is used.

“The Reynolds number is the ratio of inertial forces to viscous forces. (rehm, 2009)”

$Re = \frac{\rho v l}{\mu}$ ρ=density v=velocity l=length μ=viscosity	Diameter of a pipe can be used for L in cases of internal flow, Length of a face can be used for L in cases of external flow.
---	---

Laminar vs turbulent flow

Laminar flow is how a fluid that flows in consistent and smooth paths is described, “the fluid flows in parallel layers (connor, 2019)”. Conversely, turbulent flow is used to describe fluid that flows in a chaotic manor and “does not flow in parallel layers (connor, 2019)”. (connor, 2019)

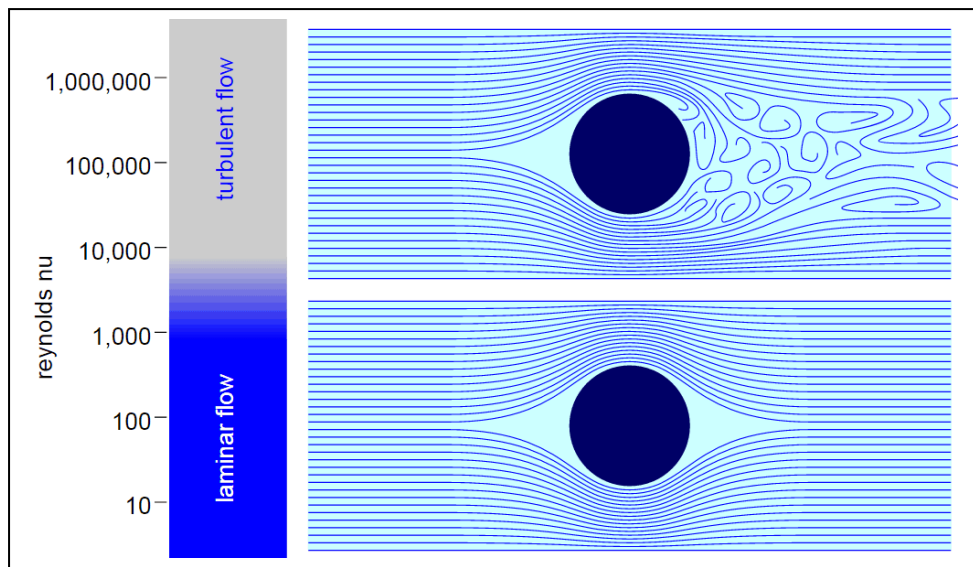


Figure 1: Laminar vs. turbulent flow (Nuclear power, 2022)

Laminar flow occurs when the Reynolds number is less than 2000.

Turbulent flow occurs when the Reynolds number is greater than 3500

Reynolds numbers that lie between 2000-4000 are known as transitional flows; in this region the flow is neither fully laminar nor turbulent. Rather a mixture of both. (connor, 2019)

To calculate the Reynolds number of the given propeller, some assumptions regarding the conditions of the propeller need to be made.

ρ =density

The fluid in question is seawater since the application of this propeller is to power a heavy-duty tanker that presumably makes voyages at sea. The density of seawater is 1025 kg/m³ at 20 degrees Celsius (Engineeringtoolbox, 2005)

20 degrees is the average temperature of the surface of seawater. Since the propeller operates relatively close to the surface of the sea I have assumed the temperature to be 20 degrees Celsius at all times. (Roemich, 2014)

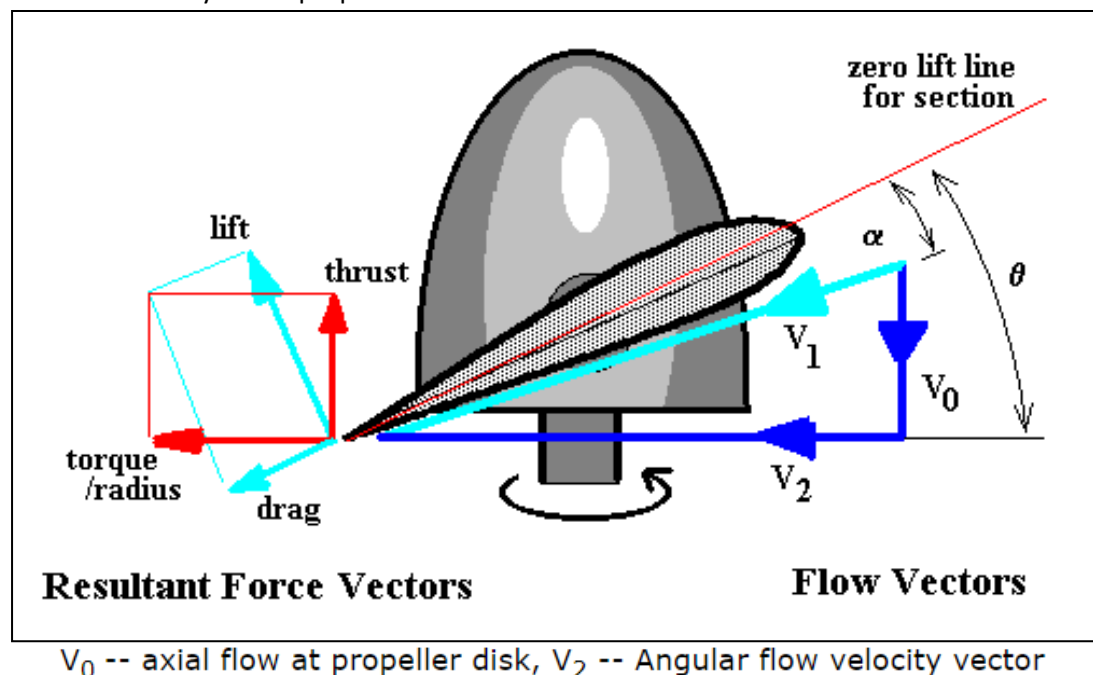
v =velocity

Two elements that contribute to the velocity of the fluid inflow.

The axial flow of the propeller disk, this is the velocity at which the tanker is moving.

The angular flow of the propeller, this is the rate at which the propeller is turning

The flow velocity of the propeller blade is the resultant of these vectors.



V_1 -- section local flow velocity vector, summation of vectors V_0 and V_2

Figure 2: Resultant Force and flow vectors of a propeller. (University of sydney, 2006)

V_2 - Rpm of typical tanker engines can range from 300 to 900 rpm. So for the purpose of this report the rpm's is assumed to be 600.

The radius of the propeller is the distance from the centrepiece to the outside of the propeller blade. As per the drawings, the length of a propeller blade is 2838.28mm.

I have assumed the centre piece of the propeller to be 750mm in diameter.

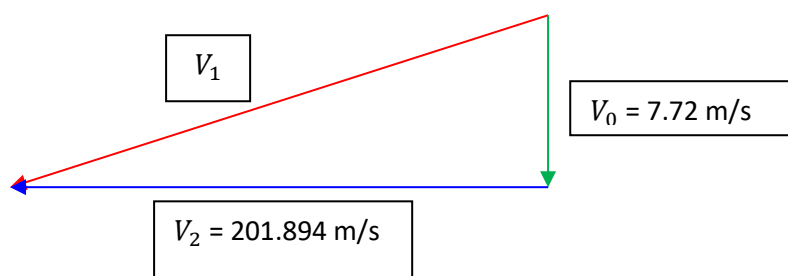
Therefore radius is $2838.25 + 750/2 = 3213.25\text{mm}$ or 3.21325m

$$v = \frac{2\pi r}{60} \times N, \text{ where } N \text{ is angular velocity (RPM)}$$

$$V_2 = 201.894 \text{ m/s}$$

V_0 -Average speed of an oil and chemical tanker is 13 to 17, so the velocity of the tanker is assumed to be 15 knots

$$15 \text{ knots} = 7.72 \text{ m/s}$$

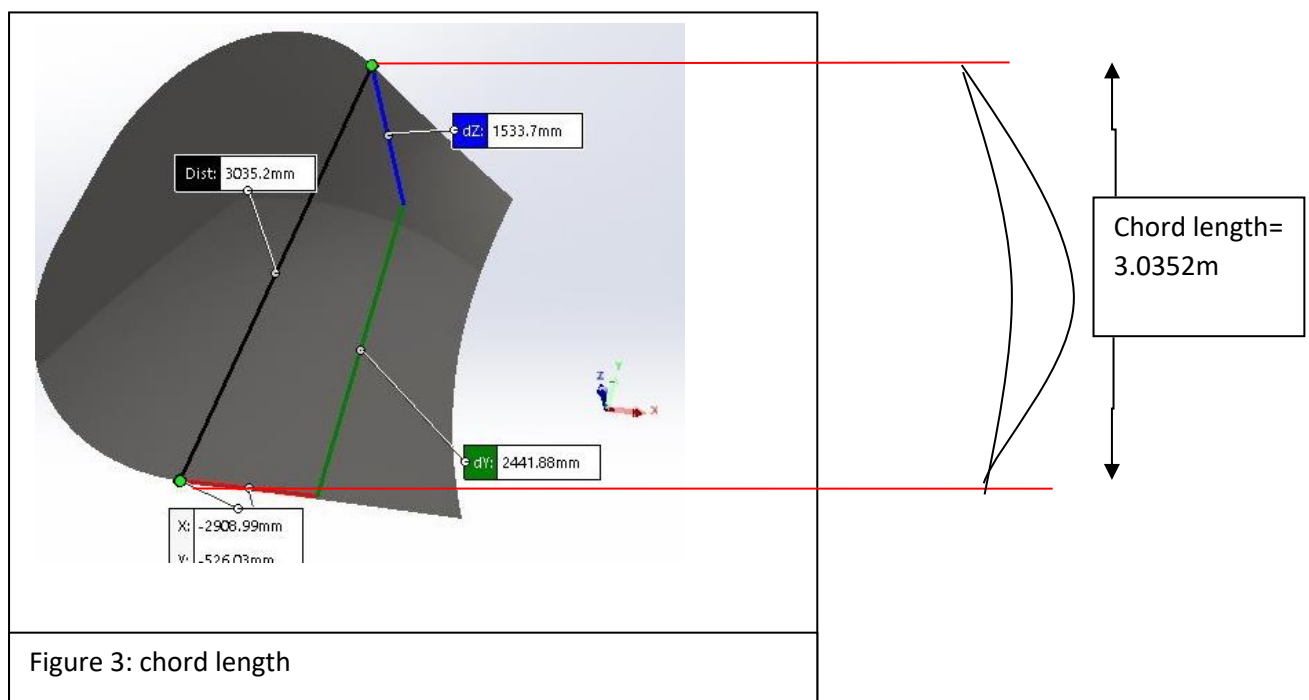


$$V_1 = \sqrt{7.72^2 + 201.894^2}$$

$$= 202.042 \text{ m/s}$$

l =length

In the case of a propeller, the critical length is the chord length of one of the blades of the propeller.



μ =viscosity

Viscosity of seawater at the presumed 20 degrees Celsius is 0.00109 ns/m^2 / pa/s (Engineeringtoolbox, 2005)

Other assumptions

I have assumed the fluid to be at rest, with a velocity of 0 m/s

Final Reynolds number

$$Re = \frac{1025 \times 202.042 \times 3.0352}{0.00109} = 576,668,647.1, \quad 5.77 \times 10^8$$

Therefore the flow is likely to be turbulent.

2.

Boundary flow

To consider the boundary flow around the propeller, a cross section of the propeller can be taken.

The propeller can be assumed to be motionless with fluid passing over it. Much like an airfoil in a wind tunnel.

When a body such as an airfoil or a propeller encounters fluid flow, a boundary layer forms around the body. This is due to the viscosity of the fluid and fluids with higher viscosities will form thicker boundary layers. (Tec science , 2020)

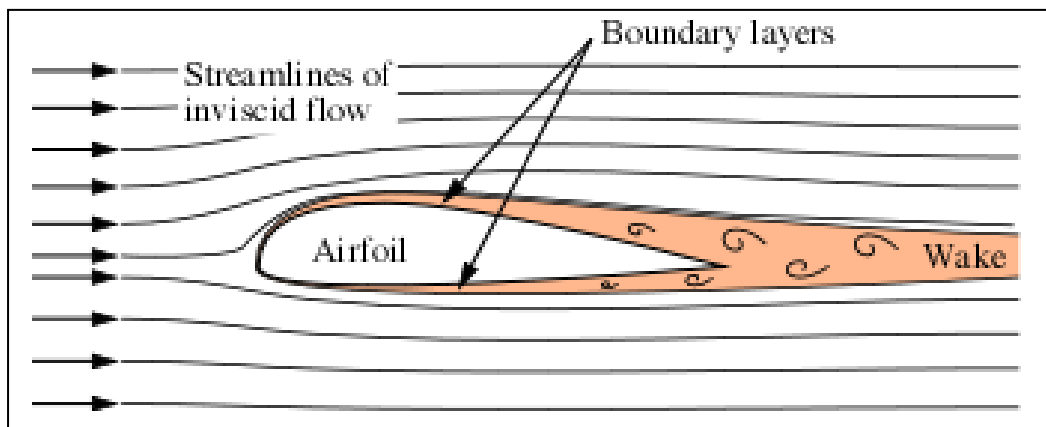


Figure 4: boundary layer over airfoil (Portland state University, 2007)

As fluid flows around the profile of the propeller, the fluid's velocity increases and therefore encounters increasingly higher pressures, as is stated by Bernoulli's principle. In some cases the flow at the boundary layer is unable to continue around the body due to the adverse pressure gradient. This causes the boundary layer fluid to be pushed backwards to the point where the fluid flow has stopped; this is called the recirculation region. The point at which the flow of the boundary layer meets the backwards flow of the recirculation is the separation point.

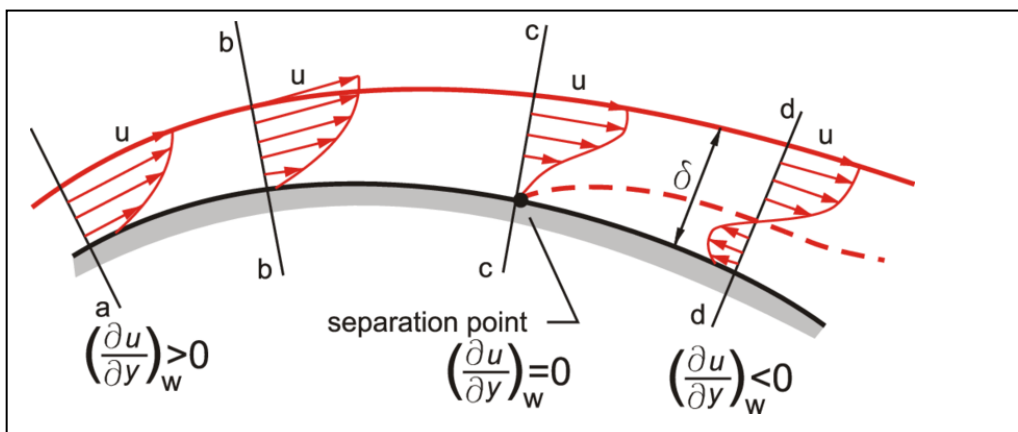


Figure 5: adverse pressure gradient (Meneghini, 2016)

After the separation from the propeller the flow can then be described as a “turbulent wake”. In this region the flow can no longer “stick” to the propeller. Therefore surface drag is now negligible, however the benefits of this drag reduction is far outweighed by the drag increase caused by the decreased pressure region caused by the wake. This means that it is within the interest of the design to minimise the magnitude of the turbulent wake.

The further along the blade the separation point, the smaller the turbulent wake and therefore the less of an effect it has on the performance of the propeller. The propeller blades will have been designed with this in mind.

Boundary layers with low Reynolds numbers are able to maintain laminar flow. Increasing the Reynolds number means that the boundary layer starts to transition to turbulent flow and then becomes turbulent. (connor, 2019)

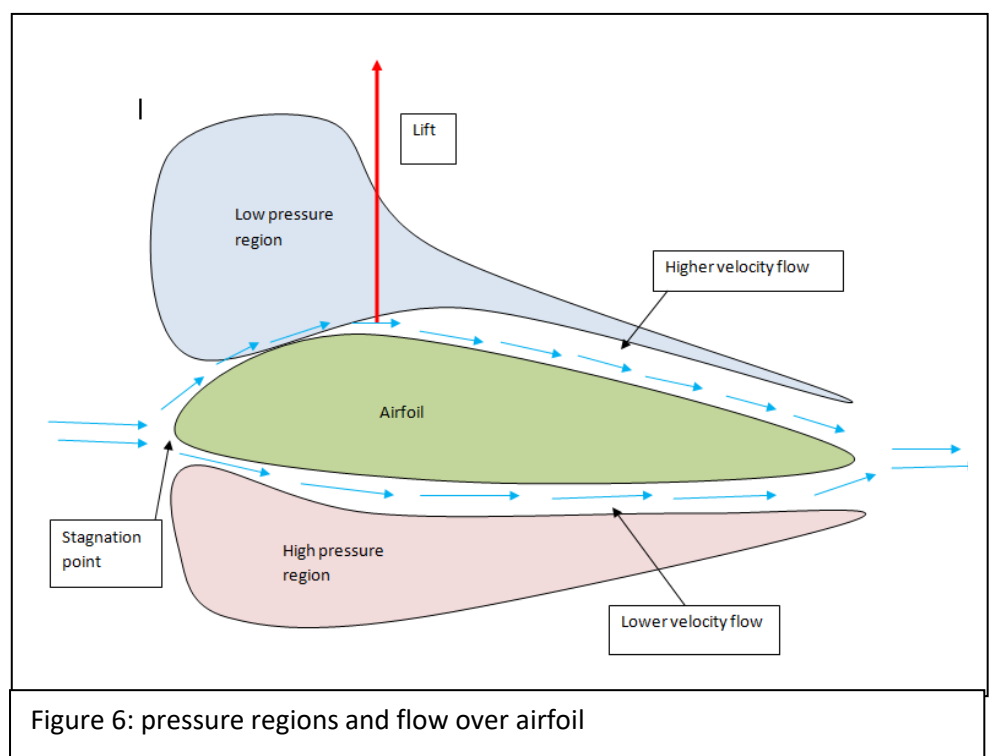
Design of propeller blade

The angle of attack of the propeller needs to be designed to accommodate the flow direction. Flow direction can be determined by considering the vectors of flow, whereby the rotational speed of the propeller and the motion of the tanker are considered. If a cross section of the propeller blade is considered the rotational speed of the propeller at this point can be calculated and the ideal angle of attack is determined.

However, at any other cross section taken of the blade, the rotational speed will be different due to the change in radius, which directly effects angular velocity. Thus, the ideal angle attach has changed. This is why the blade has been designed with a twist. To maintain the ideal angle of attack throughout the length of the blade. (Auld, 2006)

The cross section of the propeller blade will be designed with a camber i.e. asymmetrically. The implications of this are that when the fluid splits upon encountering the propeller. A portion of the fluid will travel “over” the propeller and a portion of fluid will travel “underneath”. The asymmetrical design of the blade means that the paths are not equal. The fluid travelling “above” will have a higher velocity than the fluid travelling “below”.

Bernoulli’s principle states that the differences in velocities on either side of the propeller blade will cause a region of low pressure above the blade and a region of high pressure below the blade. Thus, producing lift, which in the case of a propeller is a component of thrust.



3.

CFRPs

Carbon fibre reinforced plastics (CFRPs) are generally superior to their metallic counterparts in terms of a construction material. This is due to its fatigue resistance properties. The quality of its fatigue resistance is dependent on “the quality of interfaces between fibre and matrix. (G. Xain, 2018)” (G. Xain, 2018)

In addition to its superior fatigue properties, CFRPs exhibit other desirable properties in component design. “Their excellent specific tensile strength and modulus. (Erween A. Rahim, 2018)”. Hence the desire to use CFRPs for critical components that require good strength to weight ratios, such as propellers. (Erween A. Rahim, 2018)

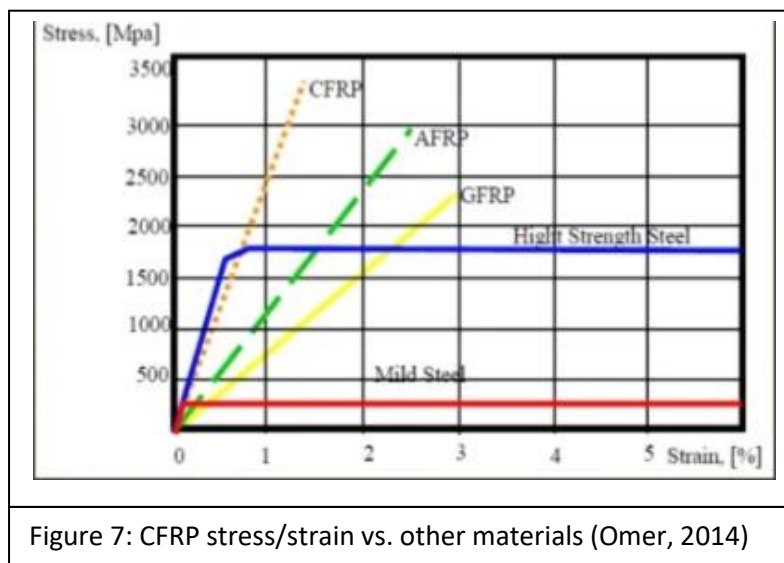


Figure 7 shows that CFRP outperforms steel as much as 5 to 10 times in terms of yielding stress. It also shows clear advantages over other composite materials such as glass fibre reinforced polymer (GFRP) and aramid fibre reinforced polymer (AFRP) (Omer, 2014)

CFRP propeller

Propellers are typically designed at an angle of attack which is optimum for a certain speed of fluid inflow. The metallic composites they are conventionally made from are able to resist deflection almost entirely; therefore the propeller is sub optimal when the vessel is travelling at any velocity other than the velocity it was designed for. (Ya-Jung Lee, 2004)

A propeller constructed from CFRP has the ability to deform under load and therefore perform optimally over a wider range of velocities. The amount of deflection can be altered by controlling the orientation of the fibre structure in the CFRP as the material exhibits different properties when load is applied in different directions. The ideal deflection can be determined through CFD (computational fluid dynamics) and the lay of the propeller can be designed accordingly. (José Pedro Blasques, 2008)

Manufacturing methods would need to be mindful of standards such as ISO 484-2:2015 Shipbuilding — Ship screw propellers — manufacturing tolerances, ensuring tolerances fall within the scope of the standard. (ISO, 2015)

Past developments

The development of materials such as CFRPs to achieve a higher performance component is part of a long development chain. The earliest propellers, such as Hookes screw propeller and Bramah's first concept of a propeller mounted onto the hull of a boat, can be attributed to the advanced components we use today. As the understanding of fluid and structural dynamics has improved, so has the quality of marine propellers. (Carlton, 2007)

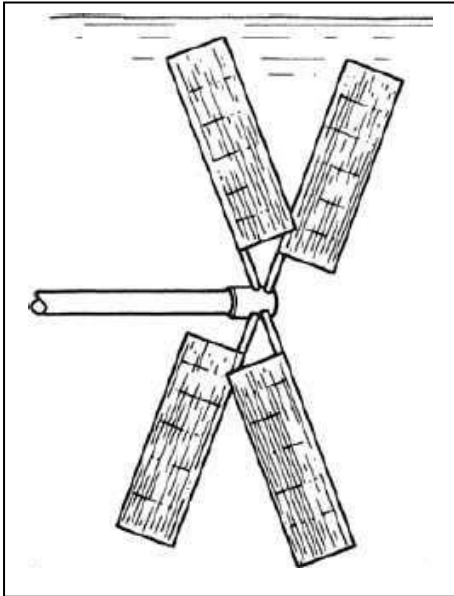


Figure 8: Hookes screw propeller (Carlton, 2007)

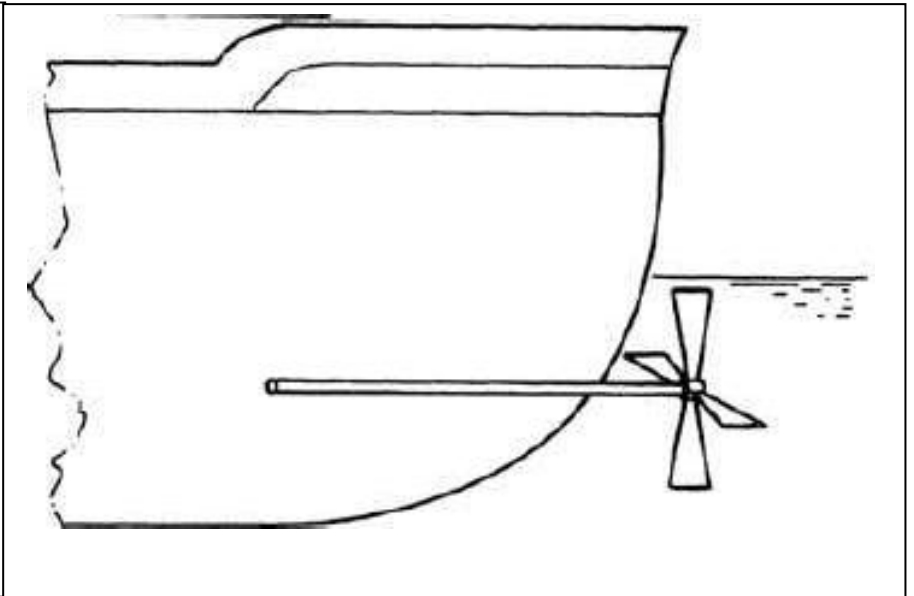


Figure 9: Bramah's screw propeller design (1785) (Carlton, 2007)

Understanding of fluid dynamics was basic until the invention of computers in the 1900's. However early pioneers of marine propellers were able to make models of their designs and test them in water. Through this data could be collected detailing how effectively the propeller performs and propels the vessel. And larger scale prototypes could then be built to assess how well the concept transfers to full scale. (Anderson Jr, 1998)

It is through this period in the 1800's that an Archimedes screw type propeller was used which later developed into a propeller comprised of multiple blades that together made up one full turn of the screw. Visually this propeller of the Napoleon (1842) (Figure 10) is not too dissimilar to the propellers we see used today. (Carlton, 2007)

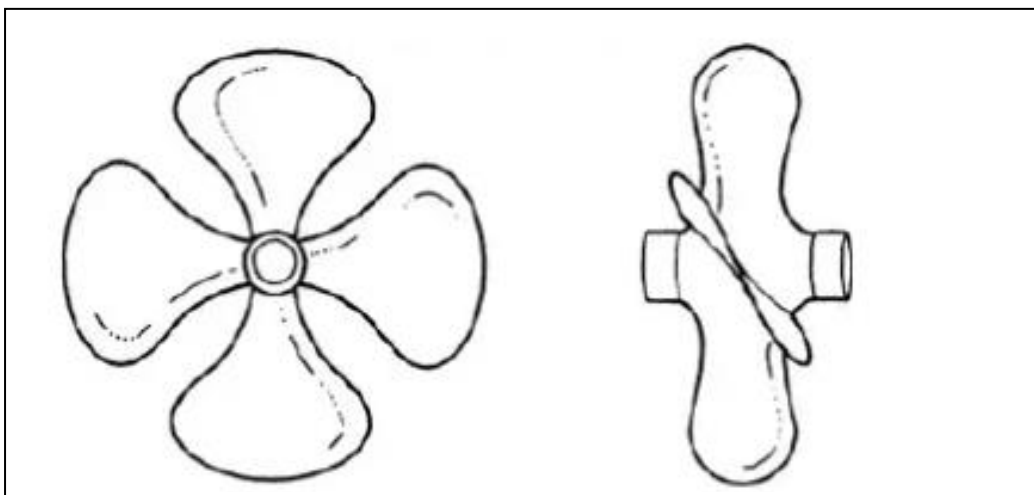


Figure 10: propeller of Napoleon (1842) (Carlton, 2007)

Current developments

With advancements in fields such as materials science, manufacturing and fluid dynamics came vast improvements of components such as propellers.

Research and development is now far more sophisticated than the trial and error approach of the 19th and 20th century. The ability to run cost and time effective simulation allows us to ask “what if?” questions and further improve our understanding of what makes components perform better.

In the case of CFPRs the question would first be “what if the component were to deflect under load?” and in discovering this would be beneficial the task is then to find an appropriate material to fulfil this proposal.

Future developments

As the capabilities of technology improves, so will the capability of CFD. This will enable us to ask more complex “what if?” questions in the future which will enable us to gain insights and make developments we are not able to make at present.

A potential area where we could see significant future developments as technology advances is through artificial intelligence (AI) and deep learning, this “provides computers with the ability to learn without being explicitly programmed (Sadrehaghighi, 2022)” (Sadrehaghighi, 2022)

Whereas current CFD practices are only capable of answering “what if?” questions and providing no insight on how to further improve the model, AI would enable us to task computers with figuring out the questions that may not be obvious to humans, and answering them accordingly.

4.

CFD

A common way to simulate fluid flow in industry is through computational fluid dynamics (CFD) whereby software is used to undertake large computations in order to predict the behaviour of fluid as it interacts with surfaces.

Bodies such as cars are designed in 3D and converted into a mesh. Creating a mesh allows the software to break down the surfaces of the physical object into manageable units that it can know the boundary conditions for. Various meshing strategies are used. A finer mesh is required for accurate results in regions of high importance; however, a fine mesh equates to more computing power required and longer times to calculate results. To limit excessive run time, coarse mesh is used in regions of low importance and the simulation boundaries are only as large as they need to be. If further increasing of the boundaries only affects the result marginally then it may not be necessary to continue refining the model at the expense of increasing the time taken to assemble the model and run the computations.

Different types of mesh shapes include

- Tetrahedral mesh
- Swept mesh
- Mapped mesh
- Triangular mesh

Among others. (Wollblad, 2018)

Known values regarding the fluid such as viscosity and density are inputted into the software.

The level of accuracy required depends on the component and its function, for example a helicopter propeller will require a far higher accuracy than an air conditioning fan, due to the catastrophic implications of inaccurately modelling a helicopter propeller. In this instance extensive time is required to ensure an appropriate method is used, moreover excessive computing times are required.

The software uses a series of Navier-stokes equations to determine the boundary conditions of each segment in the mesh. These equations consider the conservation of mass, momentum and energy. A complex CFD model can use millions of these equations to get things down to a few numbers that are useful to engineers. I.e. lift and drag.

Benefits of CFD

- Cost effective way to test and gather data for new designs.
- Design iterations can be made inexpensively. Without CFD prototypes would be manufactured and undergo physical testing in wind tunnels. Any design iterations would need redesigning, remaking of the new prototype and retesting. This process can become very costly. By using CFD new designs can be analysed easily through the software.
- Can easily simulate alternative parameters. Differencing conditions such as temperature, pressure and fluid velocity due to weather can be considered using CFD. This is difficult and expansive to do with physical testing. (Pretechnologies, 2014)

Limitations of CFD

- Excessive computing power and time required
- Poor meshing strategy and poor working methods in general could cause inaccurate results.
- Trial and error style working methods, it is not possible to run a simulation and see that for improved drag and lift the surface needs to be “X” and “Y”
- The implications of inaccurate results are costly. Products could go to market and be defective due to incorrect simulation studies. This could require things such as recalls and design changes to fix the problem. (IFES, 2014)

Mitigating limitations of CFD

In order to negate these limitations of CFD procedures such as repeating the computation from the beginning can be used to ensure the same results are achieved.

Furthermore, it would be beneficial if an independent person undertook the study to ensure the results are not influenced by bias. This can be done by a colleague or ideally an external consultant.

Once all design iterations and refinements are made, and cross examinations have been successfully complete it now may be appropriate to invest into physical testing where a prototype is built, and data is collected in a wind tunnel. By doing this the results can be physically verified and there can be enough confidence in them that products can be taken to market.

Although high costs are incurred for the physical testing, CFD has allowed this to only be necessary once per design cycle. Barring any anomalous results. Resulting in large cost savings.

Alternative methods

Navier-stoke equations is just one example of a type of physical model used in CFD, some other examples are;

“Spatial Dimension- The geometry of the inlet may be modelled in some cases using two-dimensional or axisymmetric space rather than full three-dimensional. (NASA, 2021)”

“Turbulence Models- Various algebraic, one-equation, and two-equation turbulence models exist with various parameters and free stream boundary conditions. The option of wall functions exists. (NASA, 2021)”

5.

Due to the limitations of CFD detailed in part 4, it is important to be able to verify and validate the results. Verification is essentially asking “did I solve the equations correctly?” which can be achieved by having third parties repeat the simulation and see if the results agree. (Herwig, 2006)

Validation of CFD

Validation is asking “did I solve the correct equations?” which can be achieved through physical testing and data collection. (Herwig, 2006)

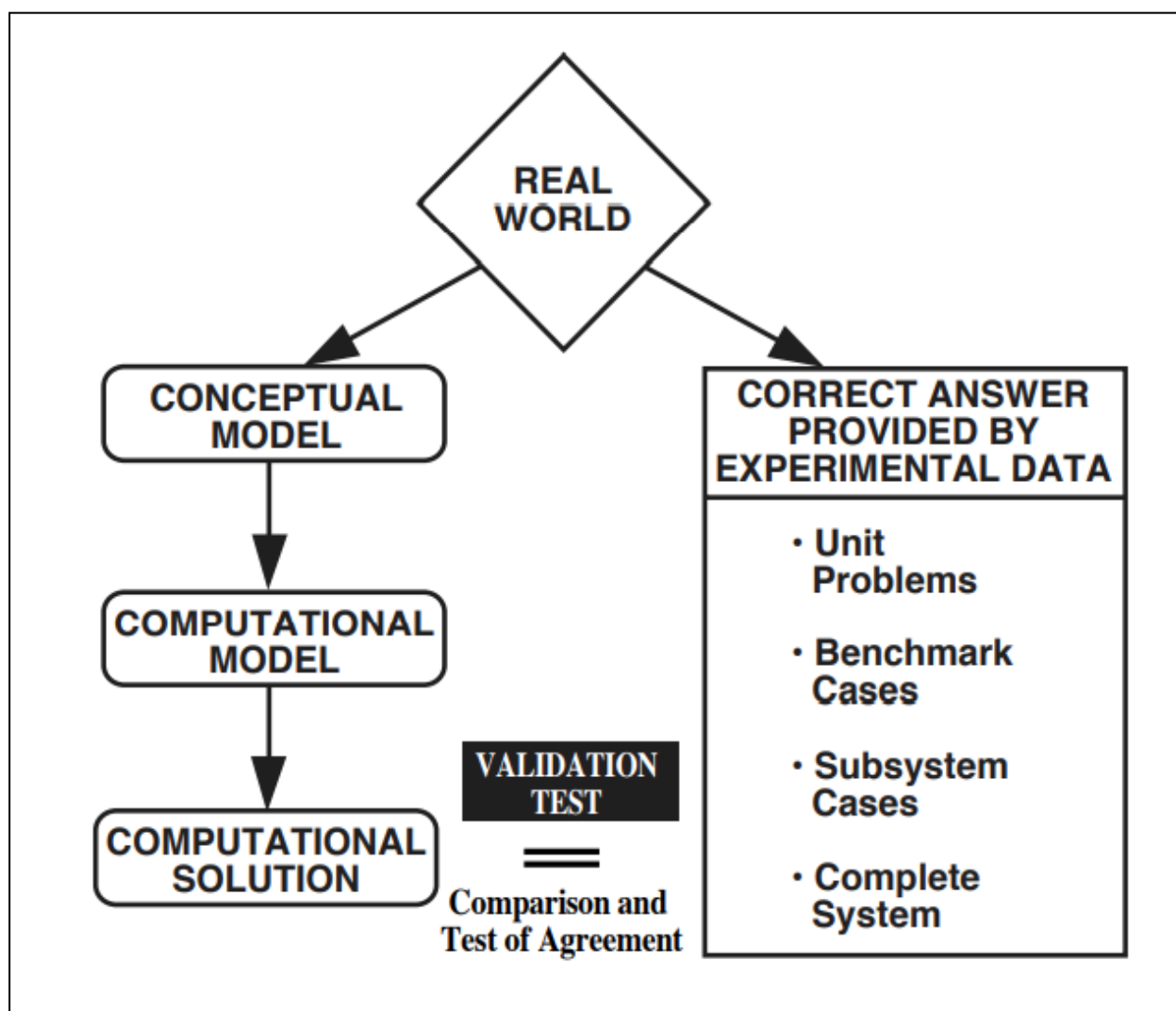


Figure 11: CFD validation procedure (Oberkampf, 2002)

Figure 11 details the validation process. The aim of this process is to identify how well the data of a physical experiment concurs with computational data; from this an understanding of the margin of error can be gained. Thus allowing us to have confidence in the results of the CFD simulations. Validation experiments do not give insight on how to improve the CFD study. (Oberkampf, 2002)

While it is not correct to assume a validation experiment is completely accurate it is fair to assume that physical testing is the best representation of reality. (Oberkampf, 2002)

Water tunnels

The field of hydrodynamics is not too dissimilar to aerodynamics. Where in aerodynamics, wind tunnels are often used to test physical models and gather data. In hydrodynamics, water tunnels are used to assess the flow of liquid that interacts with a component.

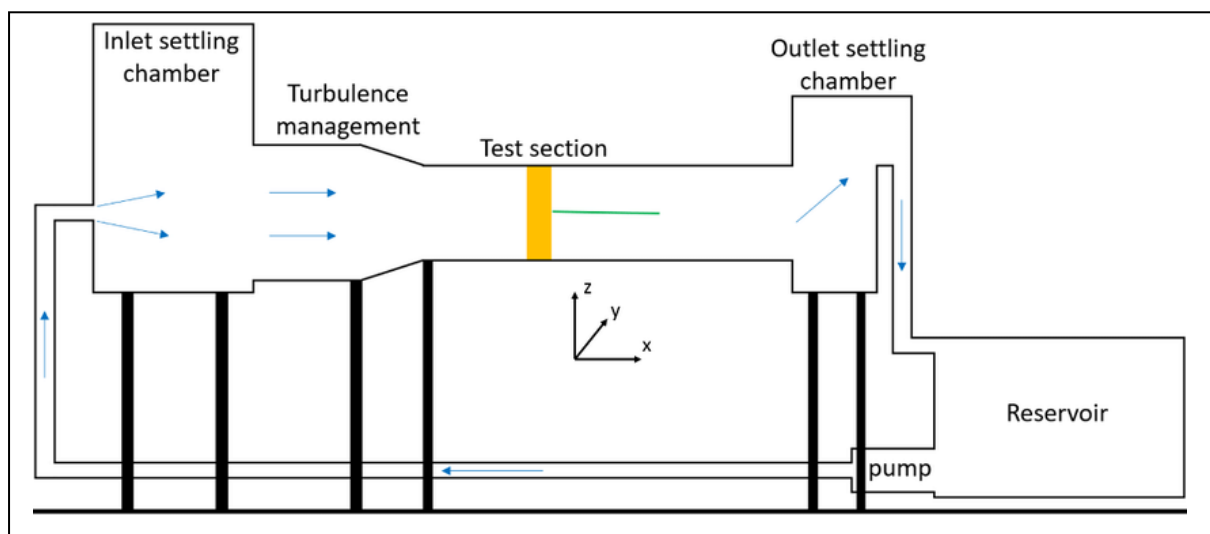


Figure 12: water tunnel schematic (Butcher, 2019)

Figure 12 details an example of the water tunnel. The fluid is circulated around the system and enters a “turbulence management” section whereby the diameter of the tunnel is reduced to ensure the flow is laminar before it travels through the test section where the prototype would be placed.

One clear limitation of the water tunnel is the size of the test section; tunnels large enough to test full scale prototypes are often extremely expensive and not readily available. To mitigate this, a scaled version of the component can be used to test.

Scaling models comes with its own set of challenges however as the conditions of the test need to be scaled also. For example the Reynolds number must be the same as the full scale. So if the prototype was built on a 2:1 scale then the velocity of the fluid would need to increase 2x in order to maintain the same Reynolds number. (McBeath, 2006)

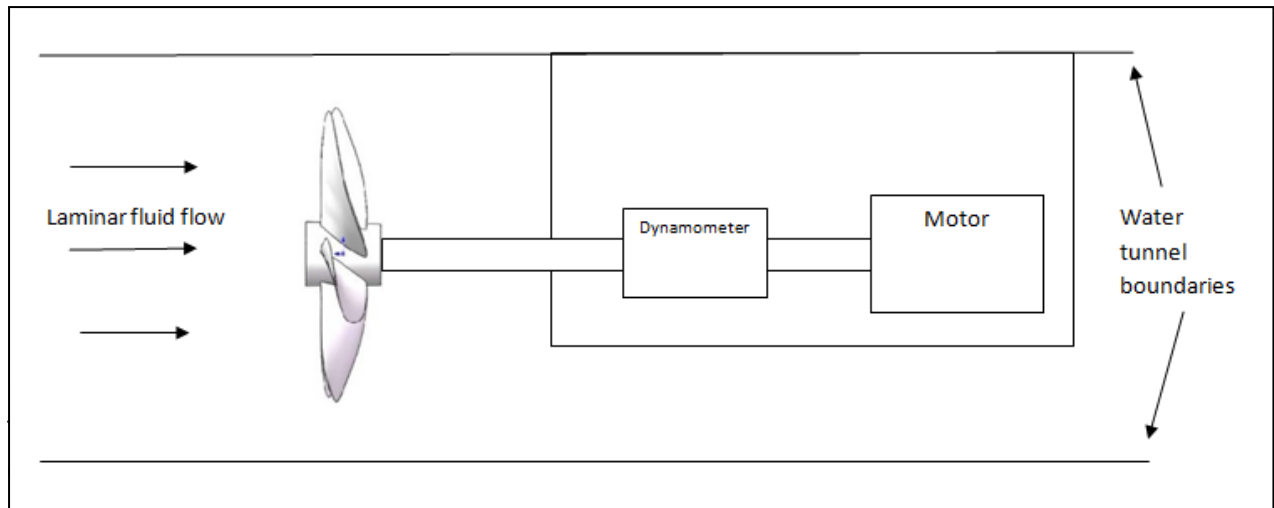


Figure 13: validation experiment schematic

Data collection

The shaft is connected to a motor that would drive the propeller much like the engine of the tanker. Also on the shaft is a Dynamometer which is a measurement instrument that gathers the thrust and torque data of the system.

The water tunnel is set up at a given fluid flow velocity and the motor is set at the desired RPM. The thrust and torque in relation to these parameters is then recorded by computers connected to the system.

A variety of fluid flow velocities and motor RPM's are tested and recorded in order to gather data at a range of speeds that the full scale product would be subjected to.

Relation to CFD

The thrust and torque data gathered from the dynamometer can be compared to the data calculated in the CFD model to determine if the values are aligned and confidence can be had in the results.

A margin of error is expected due to imperfect CFD models, imperfect physical testing and human error. A margin of 3-5% is acceptable. (Shanmugaraj, 2020)

Works Cited

Anderson Jr, J., 1998. some reflections on the history of fluid dynamics. In: R. Johnson, ed. *The handbook of fluid dynamics*. s.l.:CRC Press, pp. 2.1,2.8.

Auld, D., 2006. *Aerodynamics for students*. [Online]
Available at: http://www-mdp.eng.cam.ac.uk/web/library/enginfo/aerothermal_dvd_only/aero/propeller/prop1.html
[Accessed 23 5 2022].

Butcher, D., 2019. Cross-Correlation of POD Spatial Modes for the Separation of Stochastic Turbulence and Coherent Structures. *Fluids*, Volume 4, p. 134.

Carlton, J., 2007. *Marine propellers and propulsion*. 2nd ed. Oxford: Elsevier LTD.

connor, N., 2019. *what is laminar vs Turbulent flow*. [Online]
Available at: <https://www.thermal-engineering.org/what-is-laminar-vs-turbulent-flow-definition/>

Engineeringtoolbox, 2005. *seawater- properties*. [Online]
Available at: https://www.engineeringtoolbox.com/sea-water-properties-d_840.html

Erween A. Rahim, H. S., 2018. High performance machining of carbon fiber-reinforced plastics. In: M. T. Mohammad Jawaid, ed. *Sustainable composites for aerospace applications*. s.l.:Woodhead publishing, pp. 211-226.

G. Xain, Z. W., 2018. Carbon Fiber Reinforced Plastics – Properties. In: C. H. Z. Peter W.R. Beaumont, ed. *comprehensive composite materials*. 2nd ed. s.l.:Elsevier, pp. 342-359.

Herwig, H., 2006. *CFD: Verification and validation*. s.l.:Expert Verlag Renningen.

IFES, 2014. *Computational fluid dynamics*. [Online]
Available at: <https://www.ifes-koeln.de/en/services-20-years-project-experience/simulation-analysis/computational-fluid-dynamics-cfd.html>
[Accessed 27 May 2022].

ISO, 2015. *Shipbuilding — Ship screw propellers — Manufacturing tolerances — Part 2*. [Online]
Available at: <https://www.iso.org/obp/ui/#iso:std:iso:484:-2:ed-2:v1:en>
[Accessed 6 june 2022].

José Pedro Blasques, C. B. P. A., 2008. *Hydro-elastic tailoring and optimization of a composite marine propeller*, Lyngby: Technical university of Denmark.

McBeath, S., 2006. Tunnel vision. *Racecar Engineering*, pp. 43-48.

Meneghini, J. R., 2016. *The airfoil thickness effects on wavy leading edge phenomena at low Reynolds number regime*, s.l.: s.n.

NASA, 2021. *Overview of CFD Verification and Validation*. [Online]
Available at: <https://www.grc.nasa.gov/www/wind/valid/tutorial/overview.html>
[Accessed 5 july 2022].

Nuclear power, 2022. *Laminar vs Turbulent flow*. [Online]
Available at: <https://www.nuclear-power.com/nuclear-engineering/fluid-dynamics/flow-regime/laminar-turbulent-flow/>
[Accessed 28 june 2022].

Oberkampf, W. L., 2002. *Verification and Validation in computational fluid dynamics*. Albuquerque: Sandia National Laboratories.

Omar, M., 2014. *carbon fiber vs steel reinforcement*, Dubai: The british university in Dubai.

Omer, M., 2014. *Carbon fibre vs steel reinforcement*, Dubai: the british university in Dubai.

Portland state University, 2007. *ME 322 Lecture Notes --- Winter 2007*. [Online]

Available at: <http://web.cecs.pdx.edu/~gerry/class/ME322/notes/>

[Accessed 25 June 2022].

Pretechnologies, 2014. *advantages of computantional fluid dynamics*. [Online]

Available at: <http://www.pretechnologies.com/services/computational-fluid-dynamics/advantage>

[Accessed 27 May 2022].

rehm, B., 2009. *managed pressure drilling*. s.l.:Gulf proffesional publishing .

Roenmich, D., 2014. *VOYAGER: HOW LONG UNTIL OCEAN TEMPERATURE GOES UP A FEW MORE DEGREES?*. [Online]

Available at: [https://scripps.ucsd.edu/news/voyager-how-long-until-ocean-temperature-goes-few-more-](https://scripps.ucsd.edu/news/voyager-how-long-until-ocean-temperature-goes-few-more-degrees#:~:text=The%20average%20temperature%20of%20the,becomes%20colder%20with%20incr)

[degrees#:~:text=The%20average%20temperature%20of%20the,becomes%20colder%20with%20increasing%20depth.](https://scripps.ucsd.edu/news/voyager-how-long-until-ocean-temperature-goes-few-more-degrees#:~:text=The%20average%20temperature%20of%20the,becomes%20colder%20with%20increasing%20depth)

Sadrehaghighi, I., 2022. *Artificial Intelligence (AI) and deep learning for CFD*, s.l.: Research Gate.

Shanmugaraj, G., 2020. *What is the acceptable error variations between the CFD results and experimental data?*. Kharagpur: Indian institute of technology Kharagpur.

Taylor, J., 2006. Characterization of the effect of surface roughness and texture on fluid flow—past, present, and future. *international journal of thermal science*, 45(10), pp. 962-968.

Tec science , 2020. *flow seperation (boundary layer seperatation)*. [Online]

Available at: <https://www.tec-science.com/mechanics/gases-and-liquids/flow-separation-boundary-layer-separation/>

[Accessed 23 05 2022].

University of sydney, 2006. *Analysis of propellers*. [Online]

Available at: [http://www-](http://www-mdp.eng.cam.ac.uk/web/library/enginfo/aerothermal_dvd_only/aero/propeller/prop1.html)

[mdp.eng.cam.ac.uk/web/library/enginfo/aerothermal dvd only/aero/propeller/prop1.html](http://www-mdp.eng.cam.ac.uk/web/library/enginfo/aerothermal_dvd_only/aero/propeller/prop1.html)

[Accessed 26 june 2022].

Wollblad, C., 2018. *Your guide to meshing techniques for efficient CFD modeling*. [Online]

Available at: <https://www.comsol.com/blogs/your-guide-to-meshing-techniques-for-efficient-cfd-modeling/>

[Accessed 23 may 2022].

Ya-Jung Lee, C.-C. L., 2004. Optimized Design of Composite Propelle. *Mechanics of Advanced Materials and Structures*, 11(1), pp. 17-30.

