

Designing a tanker propellor: Fluid flow assessment

John McDyer
2010064

Module:	ACMA6010 - Structural & Fluid Analysis
Lecturer:	Dr Kelvin Lake

Table of Contents

List of Figures.....	3
Introduction	4
1. Reynolds number discussion	5-7
1.1. Reynolds number calculation.....	6-7
2. Boundary flow / layer discussion.....	8-10
3. Propellor material, performance implications and advances in design	11-12
4. Modelling techniques for maritime propellers.....	13-14
5. Experimental testing techniques	15-17
Conclusion	18
Bibliography	19-20

List of Figures

Figure 1 - Osborne Reynolds experiment	5
Figure 2 - Velocity profile	8
Figure 3- Separation of flow	9
Figure 4 - Suction side left and pressure side right.....	9
Figure 5 - Cavitation on a propellor	10
Figure 6 – CRP propellor with removeable blades for ease of maintenance	11
Figure 7 - Archimedes screw on the SS Novelty in 1840	12
Figure 8 - Modified and unmodified propellor design	12
Figure 9 - Octree mesh with different refinement levels	13
Figure 10 - Delaunay mesh with different refinement levels.....	13
Figure 11 – Cavitation tunnel schematic.....	15
Figure 12 - Vortices from experimental testing	16
Figure 13 - Measurement system	16
Figure 14 - (a) CFD analysis of cavitation (b) Experimental footage of cavitation	17

Introduction

Marine propellers haven't changed significantly throughout the years and even from the original Archimedes screw, the principal remains the same. Rotational torque is required to gain the propulsion required to move an object through the water. This report has been produced to explore the phenomenon of fluid flow around a marine propeller blade and look at modern enhancements to the traditional propeller. The report discusses the influence Reynolds number has on the boundary flow, separation and turbulence and consideration has been given to the introduction of carbon fibre reinforced plastics (CFRP) as a material selection, in particular the effect it has on the hydrodynamic properties of a ship propeller. Computational fluid analysis (CFD) is also explored with mesh techniques discussed and the link between experimental and CFD data with its advantages and disadvantages scrutinised.

1. Reynolds number discussion

The Reynolds number has a large importance within fluid flow as it plays a pivotal role in determining the patterns in a fluid's behaviour. The dimensionless Reynolds number (N_{Re}) is defined as the ratio of fluid momentum force to the viscous shear force (Guo & Ghalambor, 2005). The Reynolds number indicates whether a fluid has laminar or turbulent flow and these two conditions can be represented by a simple Osborne-Reynolds experiment consisting of a transparent pipe with a fluid stream flowing through it with the addition of a colored dye. In a laminar flow, there would be clear straight flow lines with little or no disturbances, however, if the flow rate was increased, eventually there would be evidence of eddies and disturbances which would indicate turbulent flow (Menon, 2015) as seen in Figure 1. So it can be said, "fluid flow that is slow tends to be laminar, but as it speeds up a transition occurs and it crinkles up into complicated, random turbulent flow. But even slow flow coming from a large orifice can be turbulent; this is the case with smoke stacks. Engineers and scientists don't like to say "fast" or "slow" or "small" and "big" since there is no reference. Small compared to what? Big compared to what? Since turbulence is altogether a different type of fluid flow to laminar flow, it is desirable to be able to quantify under what conditions it occurs." (Warhaft, 1998).

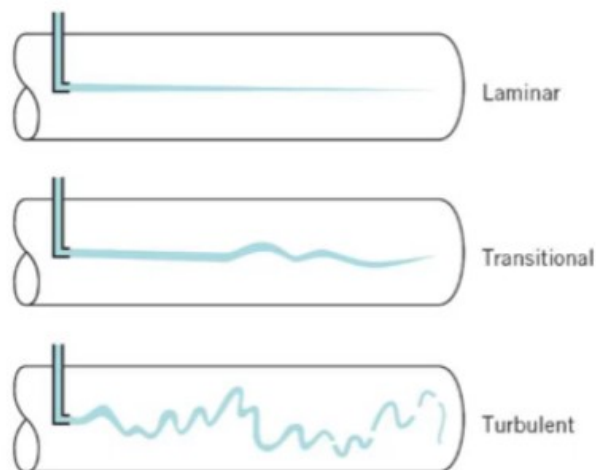


Figure 1 - Osborne Reynolds experiment

The critical Reynolds number within an enclosed system is around 2×10^3 , however, for the assignment's situation involving a propeller, we have what is known as a free flow stream where the critical Reynolds number is in the region of 1×10^5 to 1×10^6 (White, 2002). These critical numbers indicate the transition between laminar and turbulent flow, but it's worth noting that there is no single number to differentiate between the flow characteristics. This theory can also be proven by calculation whereby the flow is increased within a Reynolds calculation, the outcome of the Reynolds number clearly increases.

$$Re = \frac{\rho u l}{\eta}$$

Where:

- Re = Reynolds number
- ρ = Fluid density
- u = Fluid velocity
- l = Length scale
- η = Dynamic viscosity

Below are two examples of air flow over a flat plate at a velocity of 10 m/s and 20 m/s respectively, which highlight the change in Reynolds number. Another important characteristic to note is that the change in Reynolds number is directly proportional to the change in velocity.

$$Re = \frac{1.226 \times 10 \times 0.5}{1.784 \times 10^{-5}} = 343.61 \times 10^3$$

$$Re = \frac{1.226 \times 20 \times 0.5}{1.784 \times 10^{-5}} = 687.22 \times 10^3$$

1.1. Reynolds number calculation

To calculate the Reynolds number for the tanker propeller, we are required to work out the velocity of the water flow around the propeller i.e. idealised conditions. For this, we are assuming the ship's forward movement is a direct action of the propeller rotating and that the propeller is in open space with no turbulent incoming flow which indicates no cavitation around the propeller blades. The pitch angle of the propeller blade is assumed at 5 degrees and the diameter of the propeller at 6 meters rotating at a speed of 130 revolutions per minute.

So water speed through propeller:

$$u = \pi \times D \times \tan Pp \times \frac{rpm}{60}$$

Where: u = Velocity (m/s)
 D = Propeller diameter (m)
 Pp = Propeller blade pitch (degrees)

$$u = \pi \times 6m \times \tan 5^\circ \times \frac{130}{60}$$

$$u = 3.57m/s$$

Now if we assume that the ship is travelling through salt water at a temperature of 10°C, which would give the sea water a dynamic viscosity of 0.00141Ns/m² and a density of 1027kg/m³, we can work out the approximate Reynolds number. *N.B. All relative figures have been extracted from engineeringtoolbox.com from real world conditions that a ship may sail through.*

$$Re = \frac{\rho ul}{\eta}$$

Where: Re = Reynolds number
 ρ = Fluid density
 u = Fluid velocity
 l = Length scale
 η = Dynamic viscosity

$$Re = \frac{1027 \times 3.57 \times 3}{0.00141}$$

$$Re = 7.8 \times 10^6$$

Given the results of the approximation of the Reynolds number, it can be said that the flow around our propeller blade is turbulent, given that transition happens at approximately $1 \times 10^5 - 1 \times 10^6 Re$.

2. Boundary flow / layer discussion

To discuss the boundary flow around a marine propeller, it is important to understand the fundamentals and principals of the concept. The boundary layer occurs when the fluid flowing around an object or indeed the object through the fluid causes the molecules next to the object to 'stick' to its surface and is caused by the viscous forces of the fluid. This causes the molecules to have zero velocity at the surface of the object which in turn causes the molecules just above the surface to slow down due to collisions with the static molecules. As the distance from the surface increases there are less collisions where the molecules eventually enter the free stream again (Hall, 2021). This layer is called the boundary layer and the higher the viscosity of the fluid the bigger the boundary layer will be. This is due to the shearing forces heating the fluid which slows down the particles and dissipates their energy in the upper layers which also increases the Reynolds number. Figure 2 shows the velocity profile over a flat plate.

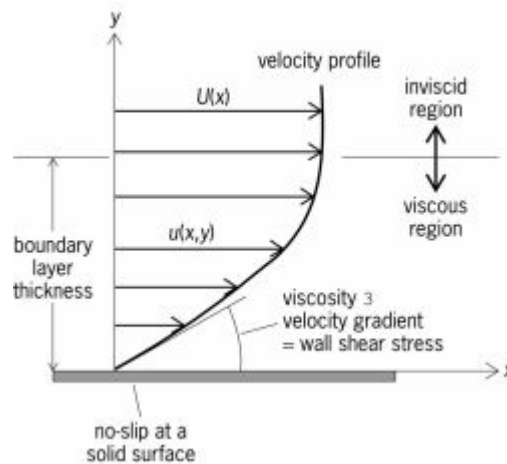


Figure 2 - Velocity profile

Now boundary layers are explained, we can relate to the assignments propeller blade and the stagnation and separation points with their positive and negative effects. The stagnation point of the propeller blade is at the leading edge of the propeller where the fluid particles come to rest and the point of maximum pressure is reached. This is relatable to Bernoulli's equation where low velocity is directly related to high pressure and vice versa. Further downstream and in line with the curvature of the propeller blade, the velocity of the flow increases and to keep an equilibrium of forces the pressure decreases. Dependent on propeller design, a separation point may occur where the curvature of the propeller blade is less effective at directing the flow in curved streamlines due to the open space downstream of the blade. Hence, the curvature in the flow reduces and the flow slows down, turning the previously favourable pressure gradient into an adverse pressure gradient of rising pressure (Anon., 2016). This then causes the low pressure molecules to move to the high pressure area causing a recirculation and a separation point as can be seen in Figure 3.

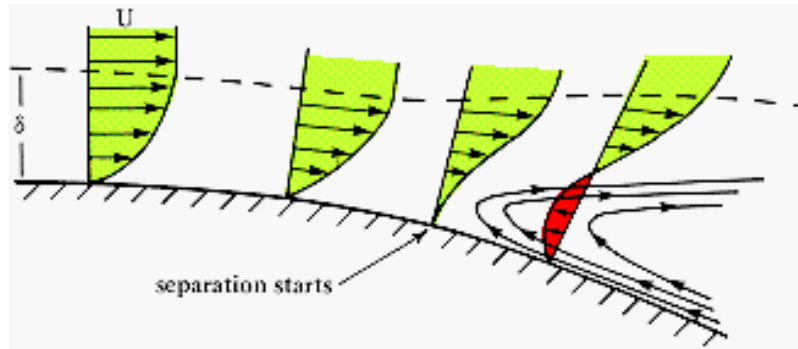


Figure 3- Separation of flow

Boundary layer separation within a ship's propeller will have its positives and negatives and it's down to the designer to ensure the right balance is achieved for the purpose of the propeller e.g. slow moving tanker or a high speed race boat. It's clear that the easier a propeller can move through the water the less energy is required to carry out the work of the rotation. Having no separation points within the propeller flow design reduces overall drag as the flow stays laminar for longer and allows for a smaller wake behind the propeller but you increase the skin friction on the propeller face which has its own negative effects in regards to drag, however, the pressure forces are usually much higher than the frictional forces. There will always be a trade off in propeller design and it is important to gain the right balance between the two and ensure the choice is suitable for the application.

Propeller designs always have a suction side and a pressure side to the blade. It becomes important to understand that turbulent or laminar flow is required on each side of the blade at certain points in order for it to work effectively. As previously explained, the turbulent flows have a higher velocity but a lower pressure and inversely for the laminar flow which explains why the difference is required for each side. It is important to have an earlier separation point on the suction side of the blade to increase velocity and reduce pressure. This allows the blade to turn with less effort and expel the water quicker at the rear of the propeller. The pressure side tends to have a later separation point and hence holds a laminar flow for longer to keep the higher pressure which in turn develops more thrust (Heinke, et al., 2019). This can be shown in Figure 3.

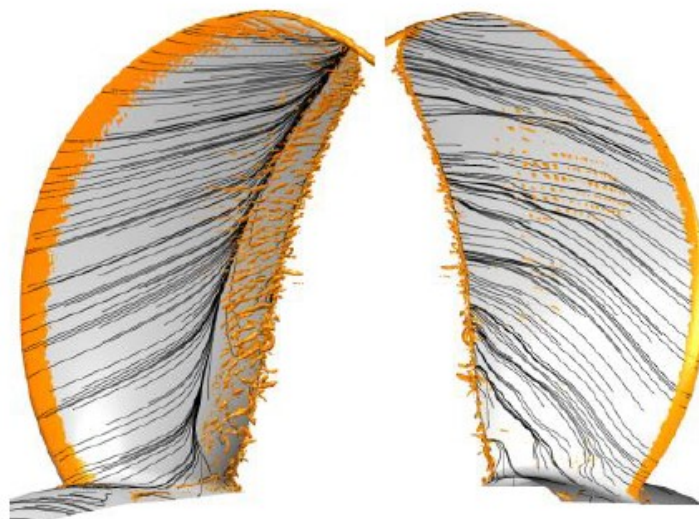


Figure 4 - Suction side left and pressure side right

The negative effect of the separation point and the higher turbulence can be a higher noise and vibration from the rotational motion and the effect of cavitation on the blade which has a detrimental effect on the propeller's life and efficiency.

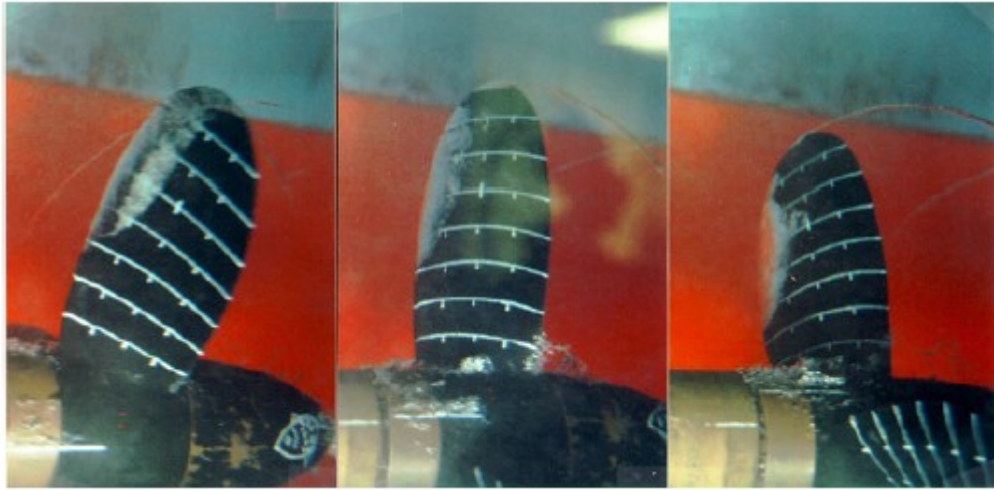


Figure 5 - Cavitation on a propellor

There is also another trade off on propeller design as where a higher rpm propeller is not typically efficient unless used on high speed craft and the larger vessels are unable to operate with a small propeller at lower rpm. At this point, the propeller size must be increased to ensure the ship has the correct torque/thrust to work at a more efficient lower rpm (Techet, 2005).

3. Propellor material, performance implications and advances in design

For the past two decades marine propellers are leaning towards composite materials as an alternative to conventionally used steels and Nickel-Aluminium-Bronze (NAB). The complex geometry of marine propellers has many challenges in design and has attained the focus of many researchers (Vardhan, et al., 2019). The use of carbon fibre reinforced plastics (CFRP) within the shipping industry is growing in popularity as manufacturing processes, designs and research become ever more present. Traditionally ship propellers have been made from metals that corrode in sea water and set up galvanic action, are easily dented, bent in groundings or strong impacts and are difficult to form into complex shapes. Plastic reinforced blades have many advantages over traditional copper based blades as it has been shown that blades are manufactured in closed moulds by a resin transfer moulding (RTM)-like process, to close tolerances so that their hydro elastic and other properties are matched which improves the capability of the propeller which in turn increases fuel efficiency and creates less noise and cavitation that traditional propellers produce. However, plastics, even when reinforced, tend to be less stiff than metals and early blades made from them were known to lose propulsive efficiency by flexing (Marsh, 2004). To eradicate this issue the latest, 'smart' contour propellers, carbon, aramid and drawn polyethylene fibres are disposed within the composite in such a way as to provide hydroelasticity. This enables the blades to react to changing load conditions by altering their pitch, so maintaining optimum propulsive efficiency across a range of throttle settings. As a result, fuel consumption can be reduced by up to 15%. In addition, adaptive self-pitching propellers reach rated rpm faster than conventional props, so that the vessel has better acceleration. Cavitation is reduced because since the blade adapts itself to different loads, the load over a given blade area tends to stay within the limits at which implosion of cavities against the blade is induced (Marsh, 2004). It's clear that CFRP marine blades have few disadvantages compared with traditional materials, however, a major discussion point for the industry is the cost of production which is why there is a tendency to use other reinforced plastics and accept the disadvantages such as less strength and damage susceptibility. Ultimately it is a cost and performance balance.



Figure 6 – CRP propellor with removeable blades for ease of maintenance

Originally the current ship propeller's predecessor was essentially known as an Archimedes screw where an elongated vane was built around a shaft. This was the design used on the SS Novelty which was produced in 1840 to test the design and is known as the first screw propelled cargo ship (Carlton, 2007).

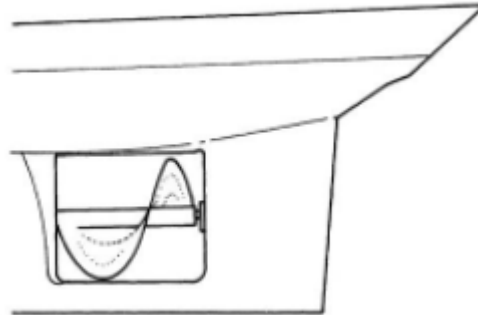


Figure 7 - Archimedes screw on the SS Novelty in 1840

Moving to more modern times and research, as with many sectors, the engineering fraternity is looking to the natural world for improvements and marine engineers are no different in terms of propeller design. It has been noticed that humpback whales had tubercle shapes on the front of their pectoral fins and has since undergone varying levels of analysis and testing, however, the results are not consistent. It is stated that tubercles introduce extra separation on the propeller blades which has in turn caused a higher turbulence around the blade which has consequently reduced torque performance and increased vibration (Stark & Shi, 2021) and the published article on CFD letters shows the conclusion to remain the same where it states "Overall, the tubercle-like shape of the propeller has a significant effect on the flow on the surface, which tends to affect the propeller performance" (Arifin, et al., 2022). However, another paper suggests that "the results indicate that while there is a substantial increase in thrust generation of up to 10%, there is also a 5% decrease in propeller efficiency" (Ibrahim & New, 2015). Overall there does appear to be a consistent drop in efficiency where in some cases an increase of torque and thrust occurs. There is still a large amount of research into modifying propeller blades in the tubercle way but more design analysis is required.

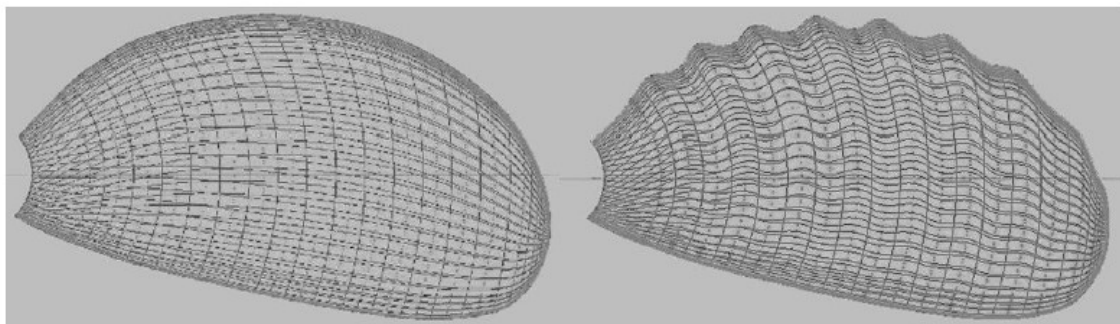


Figure 8 - Modified and unmodified propellor design

4. Modelling techniques for maritime propellers

When considering modelling techniques used for analysing fluid flow around a propeller, the immediate thought is computational fluid dynamics (CFD). CFD is an extremely useful tool in fluid dynamics especially where complex shapes such as a propeller blade requires an impossible amount of calculations to observe any real world outcomes. At best, we can approximate using only a certain number of vague points. With the advance in computational power, CFD has become a reality in the design and analysis propeller blades (Anderson, et al., 2008). CFD carries out an enormous amount of calculations on ever changing scale due to each mesh changing in respect to time. This is where we can look at modelling/mesh techniques for getting the best outcome for the calculations. The first technique we can consider is the Octree mesh as can be seen in Figure 9 where it creates a large cube around the geometry and splits this over and over into smaller tetrahedral elements until it reaches the mesh requirements. Then it projects elements on the geometry. The elements outside the geometry are discarded. The second Delaunay method first creates a mesh on the surfaces of your geometry. From there it will start to grow the mesh inside your geometry, until it is completely filled with elements as can be seen in Figure 10.

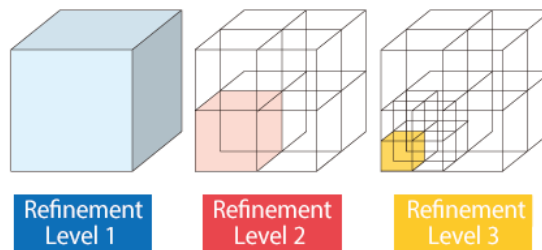


Figure 9 - Octree mesh with different refinement levels

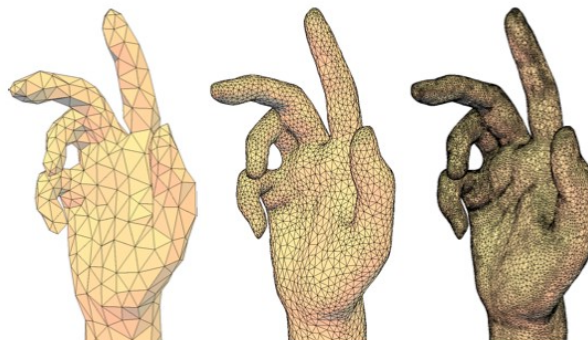


Figure 10 - Delaunay mesh with different refinement levels

There are a number of limitations to CFD which start with cost; CFD software licenses and the computing power required can be extremely expensive to purchase. Another limitation is with the effect of individual components that affect the incoming/outgoing flow, for instance, a ship hull creating turbulence for the propeller and the rudder receiving the flow/vortex i.e. interlinking more components. The number of calculations required and the variables make these figures loose approximations at best which has been proven by open water testing (Darie, 2012). A severe limiting factor is the capability of the engineer using the software with regards to set up and understanding the underlying calculation to verify results. It would be easy for an inexperienced engineer to produce data that does not match real works situations. Lastly the CFD software cannot reproduce physics that are not properly included in the formulation of the problem such as turbulence. Most CFD calculations of turbulent flows now contain

turbulence models which are just approximations of the real physics, and depend on empirical data for various constants (Anderson, et al., 2008). Even with the limitations discussed, CFD is an extremely useful tool within the design and analysis of any object, it can identify issues and gain understanding of a design before prototypes are produced and real world testing begins where both can be extremely expensive scenarios.

5. Experimental testing techniques

The testing of a propeller and its surrounding characteristics is generally carried out using open water tests. This can either be via a towing tank or a cavitation tunnel as shown in Figure 11 which is the chosen test to be carried out. The main reason for this is that towing tanks need to be reset after each run and data gathering could require multiple runs whereas the cavitation tunnel has a continuous flow with no need to reset the tests, its more dynamic in a sense. The setup of the experiments should replicate the CFD model to ensure that the results can be correlated to the initial CFD analysis. To understand why these tests are called open water tests an explanation of the cavitation tunnel is required. "A cavitation tunnel is a closed channel in the vertical plane recirculating water by means of an impeller in the lower horizontal part. This way the high hydrostatic pressure ensures that even for reduced pressure in the tunnel, the impeller itself will not cavitate. The actual test section is in the upper horizontal part. The test section is provided with observation glass ports. The tunnels are designed to give (almost) uniform flow as inflow to the test section. If just the propeller is tested (with the driving shaft downstream as the system we are considering), it is effectively tested in open water" (Bertram, 2012).

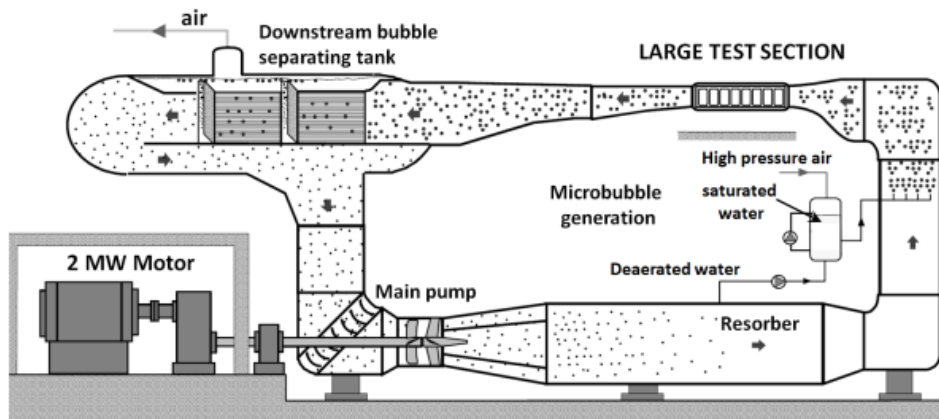


Figure 11 – Cavitation tunnel schematic

The setup of the propeller should have uninterrupted flow as per the CFD model and should be upstream to its mounting device as explained above. This will ensure laminar flow enters the vanes of the propeller giving accurate results relating to propeller performance only. The propeller will then be rotated at the required RPM and the correct water velocity supplied by the cavitation tunnel (King Propulsion, 2018). If testing was required which considered a ship's disturbance, then grids can be placed upstream to generate the wake of the ship. However this requires considerable experience and results are still not consistent. The International Towing Tank Conference has produced specific guidelines for the testing conditions and uncertainty calculations for cavitation tunnels. In particular document 7.5-02-03-02.1 which covers open water testing (Quality Systems Group of the 28th ITTC, 2014).

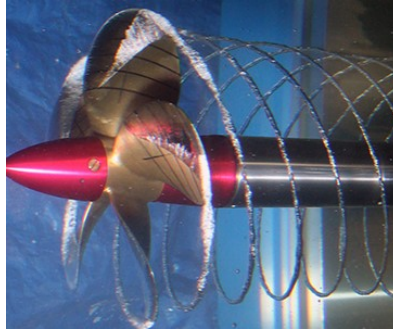


Figure 12 - Vortices from experimental testing

The results that are primarily obtained from the tests are evidence of cavitation/flow at a range of RPM, water velocity and the thrust/torque figures. The thrust/torque & RPM is taken by means of a dynamometer and data logged. See a typical measurement system in Figure 13.

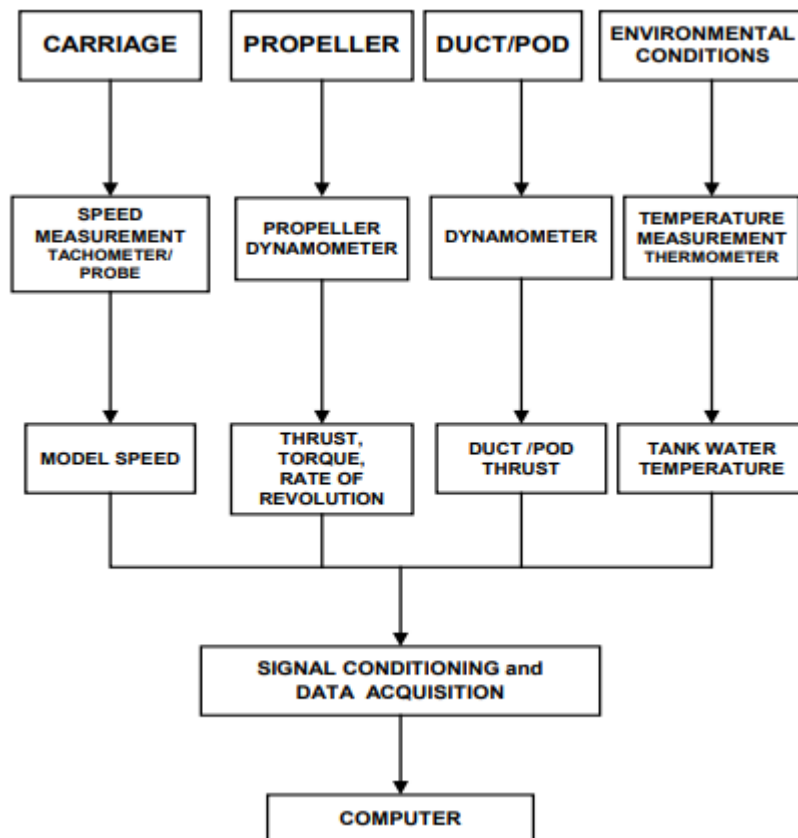


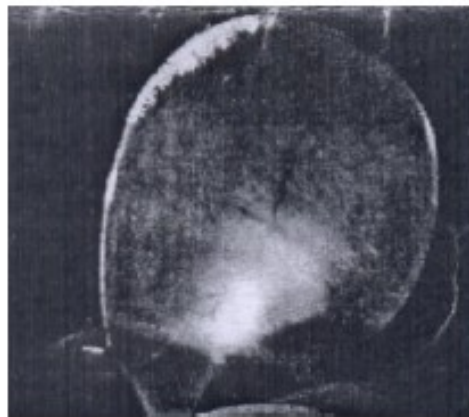
Figure 13 - Measurement system

The cavitation/flow is captured using a stroboscope that illuminates the propeller intermittently such that propeller blades are always seen at the same position. The eye then perceives the propeller and cavitation patterns on each blade as stationary (Bertram, 2012). This is then recorded by high speed videography. Any water pressures required can be monitored via pitot tubes to ensure the system is operating correctly. Once the data is scrutinized it can be put into an open water diagram to find the optimum efficiency using the J factor accounting for any discrepancies. Using the known values for propeller RPM and water velocity the data can then be correlated

against the CFD data to identify any discrepancies or give confidence in the model. A comparison of cavitation data between CFD and experimental data can be seen in Figure 14.



(a)



(b)

Figure 14 - (a) CFD analysis of cavitation (b) Experimental footage of cavitation

Conclusion

Propeller design is a wide ranging topic to understand due to the many variables that come along with it. These include differing applications, turbulence from vessel wake and material selection among others.

Propeller design will always have a trade off in some way, whether that's size to thrust ratio or a material to blade distortion amount. Even with the consistent advances in CFD, engineers and designers are unable to fully understand the turbulent flows within any given system, however, the link between experimental data and CFD has shown to be very close. In conclusion, propeller design still has much research scope, in particular with more exotic materials and processing coming to market.

Bibliography

Anderson, J. et al., 2008. *Computational Fluid Dynamics - An introduction*. 3rd ed. Berlin: Springer.

Anon., 2016. *Aerospace Engineering*. [Online]
Available at: <https://aerospaceengineeringblog.com/boundary-layer-separation-and-pressure-drag/>
[Accessed 26 May 2022].

Arifin, M. D., Felayati, F. M. & Muhammad, A. H., 2022. Flow Separation Evaluation on Tubercle Ship Propeller. *CFD letters*, 14(4), pp. 43-50.

Bertram, V., 2012. *Practical Ship Hydrodynamics*. 2nd ed. Oxford: Butterworth Heinemann.

Boucheron, R., Amailland, S. & Thomas, J.-H., 2017. *Experimental modal decomposition of acoustic field in cavitation tunnel with square*. Boston, Acoustical Society of America.

Carlton, J., 2007. *Marine Propellers and Propulsion*. 4th ed. Oxford: Elsevier.

Darie, B., 2012. *Numerical simulation of the flow field around a*. Liege: s.n.

Fish, F. E. & Battle, J. M., 1995. Hydrodynamic design of the humpback whale flipper. *Journal of Morphology*, 225(1), pp. 51-60.

Guo, B. & Ghalambor, A., 2005. *Natural Gas Engineering Handbook*. 2nd ed. Houston: Gulf Publishing Company.

Hall, N., 2021. *nasa.gov*. [Online]
Available at: <https://www.grc.nasa.gov/www/k-12/airplane/boundlay.html>
[Accessed 26 May 2022].

Heinke, H. J., Hellwig-Rieck, K. & Lubke, L., 2019. Influence of the Reynolds Number on the Open Water Characteristics of Propellers with Short Chord Lengths. *Sixth International Symposium on Marine Propulsors*.

Ibrahim, I. H. & New, T. H., 2015. *Tubercle Modifications in Marine Propeller Blades*. s.l., s.n.

Khan, M. A., Uddin, K. S. & Ahmed, B., 2013. Design and Dynamic analysis on composite propeller of ship using FEA. *International Journal of Advanced Trends in Computer Science and Engineering*, 2(1), pp. 310-315.

Kim, J.-H., Kim, H.-G., Lee, B.-C. & Im, S., 2003. Adaptive mesh generation by bubble packing method. *Structural Engineering and Mechanics*, 15(1), pp. 135-149.
King Propulsion, 2018. *Open Water Propeller Tests*. s.l.:s.n.

Marsh, G., 2004. A new start for marine propellers?. *Reinforced Plastics*, 48(11), pp. 34-38.

Menon, E. S., 2015. Chapter Five - Fluid Flow in Pipes. In: *Transmission Pipeline Calculations and Simulations Manual*. Houston: Gulf Publishing Company, pp. 149-234.

Quality Systems Group of the 28th ITTC, 2014. *Recommended Procedures & Guidelines - Open water test*. Zurich, International Towing Tank Conference.

Stark, C. & Shi, W., 2021. *The Influence of Leading-Edge Tubercles on the Sheet Cavitation Development of a Benchmark Marine Propeller*. New York, The American Society of Mechanical Engineers.

Techet, P. A. H., 2005. *Hydrodynamics*, s.l.: s.n.

Vardhan, D. H., Ramesh, A. & Chandra Mohan Reddy, D., 2019. *A Review On Materials Used For Marine Propellers*. Andhra Pradesh, Elsevier.

Warhaft, Z., 1998. *An Introduction to Thermal Fluid Engineering - The Engine and the Atmosphere*. 1st ed. Cambridge: Cambridge University Press.

White, F. M., 2002. *Fluid Mechanics*. 4th ed. Boston: McGraw-Hill Higher Education.