

Awais Ahmed Student ID - 2012416

Assignment 1

Structural and Fluid Analysis (ACMA6010)

Abstract

In section 2.0 the Importance of Reynolds number and how it influences fluid flow was identified and discussed. Calculations were performed to give a Reynolds number for a given propeller geometry.

Fluid flow around propeller blades can vary based on a multitude of factors and design considerations which are discussed in section 3.0.

Comparison of CFRP to materials that were used historically, currently and will be used in the future of propeller development was discussed in section 4.0 this section also includes relevant industry standards and codes of practice.

Section 5.0 builds upon knowledge of multiple modelling techniques and identifies the advantages and disadvantages of each technique.

In section 6.0 a PIV experiment was discussed to compare to a CFD simulations. Additions to the PIV experiment to overcome inherent disadvantages were described. Examples of PIV experiment results were shown and alterations to a prototype model to improve cost effectiveness was also discussed.

Key Words
CFD – Computational Fluid Dynamics
CFRP – Carbon-fibre-reinforced polymer
FDM – finite-difference method
FEA - finite-element analysis
FEM - finite-element method
FVM - Finite Volume Method
GFRP – Glass-fibre-reinforced polymer
ISO - International Organization for Standardization
NAB - nickel-aluminium-bronze
PIV - Particle image velocimetry

Table of Contents

	Page Number
List of Figures	4
List of Tables	5
1.0 Introduction	6
2.0 Task 1	7-8
2.1 Importance of Reynolds Number	7
2.2 Calculation of Reynolds Number for Provided Propeller Dimensions	8
3.0 Task 2	9-13
3.1 Description of Typical Boundary Layer	9
3.2 How Reynolds Number affects the Boundary Layer	10-12
3.3 How Propeller Design affects Performance	13
4.0 Task 3	14-18
4.1 CFRPs Implications on Propeller Performance	14
4.2 High Tensile Brass	15
4.3 Discussion and Comparison of CFRP and NAB	16
4.4 Discussion and Comparison of CFRP and GFRP	17
4.5 Relevant Industry Standards and Codes of Practice	18
5.0 Task 4	19-22
5.1 Advantages and Disadvantages of CFD	19
5.2 Advantages and Disadvantages of FEA/FEM	20
5.3 Advantages and Disadvantages of FVM	21
5.4 Advantages and Disadvantages of FDM	22
6.0 Task 5	23-28
6.1 Experiment Introduction	23
6.2 PIV Experiment Set-up and Testing	24-25
6.3 Examples of PIV Experiment Results	26-27
6.4 Cost Saving Method	28
7.0 Conclusion	29
8.0 Works Cited	30-34
9.0 Appendix A – CFD Simulation for PIV Testing	35
10.0 Appendix B	36
10.1 Displacement ratio consideration for small scale model	36
10.2 Reynolds Number consideration for small scale model	36

List of Figures

	Page Number
Figure 1 – Transition of flow across a plate (MIT News Office, 2020)	7
Figure 2 – Estimated Reynolds Number Calculation	8
Figure 3 – Representation of boundary layer on suction side of propeller (Rijpkema, Baltazar and Falcão de Campos, 2016 p.2)	9
Figure 4: “Contour line of total pressure loss ($C_{pl} = 0.01$) representing the boundary-layer shape for $J=0.7$ for E779A propeller at $0.7R$ ” (Rijpkema, Baltazar and Falcão de Campos, 2016 p.8)	10
Figure 5: “Contour plot of eddy-viscosity ratio (contour line at $\mu_e/\mu = 2$) for $J=0.7$ for E779A propeller at $0.7R$ ” (Rijpkema, Baltazar and Falcão de Campos, 2016 p.10)	11
Figure 6: Diagram showing turbulence present at the trailing edge	12
Figure 7: Image depicting how local angle of attack affects vectors (Aerodynamics4students, 2019)	13
Figure 8: PIV Experiment Set-up (Overhead View) (Nyongesa et al., 2022)	24
Figure 9: Typical region of interest size for CFD (a) and PIV experiment (b) (Nyongesa et al., 2022, p.12)	26
Figure 10: Velocity Magnitude colour Map for CFD (a) and PIV experiment (b) (Nyongesa et al., 2022, p.12)	26
Figure 11: Example of Velocity Streamline Result (Nyongesa et al., 2022, p.16)	27
Figure 12: Example of Propeller Pressure Distribution Result (Nyongesa et al., 2022, p.18)	27
Figure 13: 3D model of the PIV Test Tank (Nyongesa et al., 2022, p.4)	35
Figure 14: Displacement Ratio Formula (Hara et al., 2015)	36
Figure 15: Bending Elastic Moment Formula (Hara et al., 2015)	36

List of Tables

	Page Number
Table 1 – Relevant Codes of Practice	18
Table 2 – Relevant industry Standards	18
Table 3 – Advantages and Disadvantages of CFD	19
Table 4 – Advantages and Disadvantages of FEA/FEM	20
Table 5 – Advantages and Disadvantages of FVM	21
Table 6 – Advantages and Disadvantages of FDM	22

1.0 Introduction

Fluid flow is an essential factor for engineers to consider in designing relevant products. It is a very complex process with many contributing factors that affect the way it behaves. This report discusses the importance of relevant terms used in fluid analysis and covers factors that influence fluid flow and how design considerations can affect the performance of ships. A comparison of propeller materials and their effect on performance will be discussed as well as the prototyping of propellers and validation of CFD models through physical experimentation. CFD models are essential for prototyping and cost-saving, and there is a multitude of modelling techniques that can be used, some of which will be discussed within this report.

2.0 Task 1

2.1 Importance of Reynolds Number

Reynolds number is a dimensionless number that aids in flow behaviour prediction.

Laminar flow is indicated by a low Reynolds number, while a high Reynolds number indicates turbulent flow.

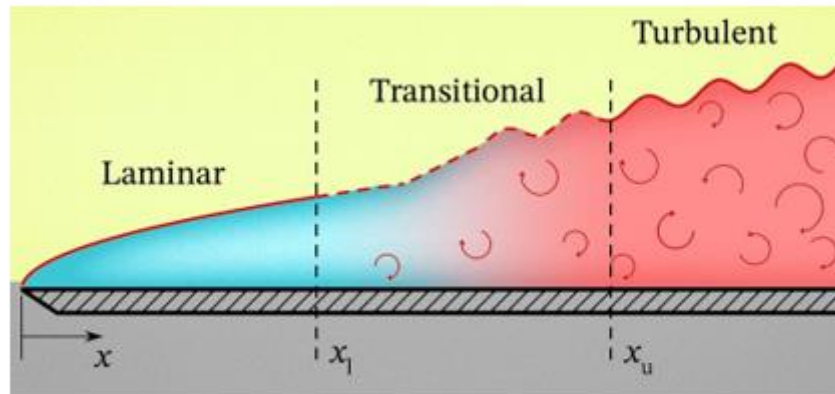


Figure 1 – Transition of flow across a plate (MIT News Office, 2020)

The identification of flow regimes using Reynolds numbers aids in the development of an ideal flow model for efficient fluid system design (Cadence CFD, 2022).

The nature of the fluid flow within an aerodynamic or hydraulic system is vital knowledge for the design of the system. Because different flow patterns can impose different forces within the system. Engineers and system designers can study the challenges that turbulent and laminar flow patterns cause and include these considerations when evaluating system performance by determining whether the flow is turbulent or laminar, thus signifying the importance of the Reynolds number (Cadence CFD, 2022).

It is possible to optimise the design of fluid-dependent systems and assure efficient performance by employing an advanced CFD model for modelling flow behaviour. The simulation method can be simplified by using Reynolds number, which can capture all laminar and turbulent motion and boundaries for effective mesh production by utilising flow prediction, further signifying the importance of Reynolds number (Cadence CFD, 2022).

Flow is more resistant to separation at higher Reynolds numbers as atoms have more energy causing them to convect with greater speed with respect to mean flow. Greater mixing is one cause for this, but increased convection velocity has also contributed significantly to increased sweep events (evidenced by a positive skewness factor). As a result, there is more velocity transfer, which leads to an increase in skin friction. The amplitude modulation process is responsible for a rising momentum with Reynolds number, and as it becomes stronger, the mean velocity profile approaches that of the convection velocity. The effect of increased momentum was confirmed to be a natural mechanism of flow stabilisation, functioning similarly to the drag crisis effect, which delays separation with a rise in the Reynolds number (Drózdź, Niegodajew, Romańczyk and Elsner, 2021).

2.2 Calculation of Reynolds Number for Provided Propeller Dimensions

Reynolds Number Calculation

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

Annotations for the formula:
- ρ : density
- v : velocity
- L : critical length
- μ : dynamic viscosity of fluid

Assumptions Made

- * Average Sea temperature of 10°C , giving an absolute dynamic viscosity of 0.00141 .
- * Density of sea water is 1026 kg/m^3
- * Typical RPM of tanker ship to be 100 RPM .
- * Propeller shaft diameter of 500 mm .

The propeller image shows a length of 2838.28 , combining the length of two blades gives 5676.56 .

Total diameter of propeller including shaft gives $= 500 + 5676.56 \text{ mm} = 6.177 \text{ m}$

Velocity $= 100 \text{ RPM} / 3.0885^{\text{m}} (\text{radius}) = 32.34 \text{ m/s}$

Reynolds number $= \frac{1026 \times 32.34 \times 6.177}{0.00141} = 145.36 \times 10^6$

Value indicates that flow is very turbulent.

Figure 2 – Estimated Reynolds Number Calculation

3.0 Task 2

3.1 Description of Typical Boundary Layer

Below is a labelled illustration of a typical boundary layer on the suction side of a propeller blade at a model scale Reynolds number.

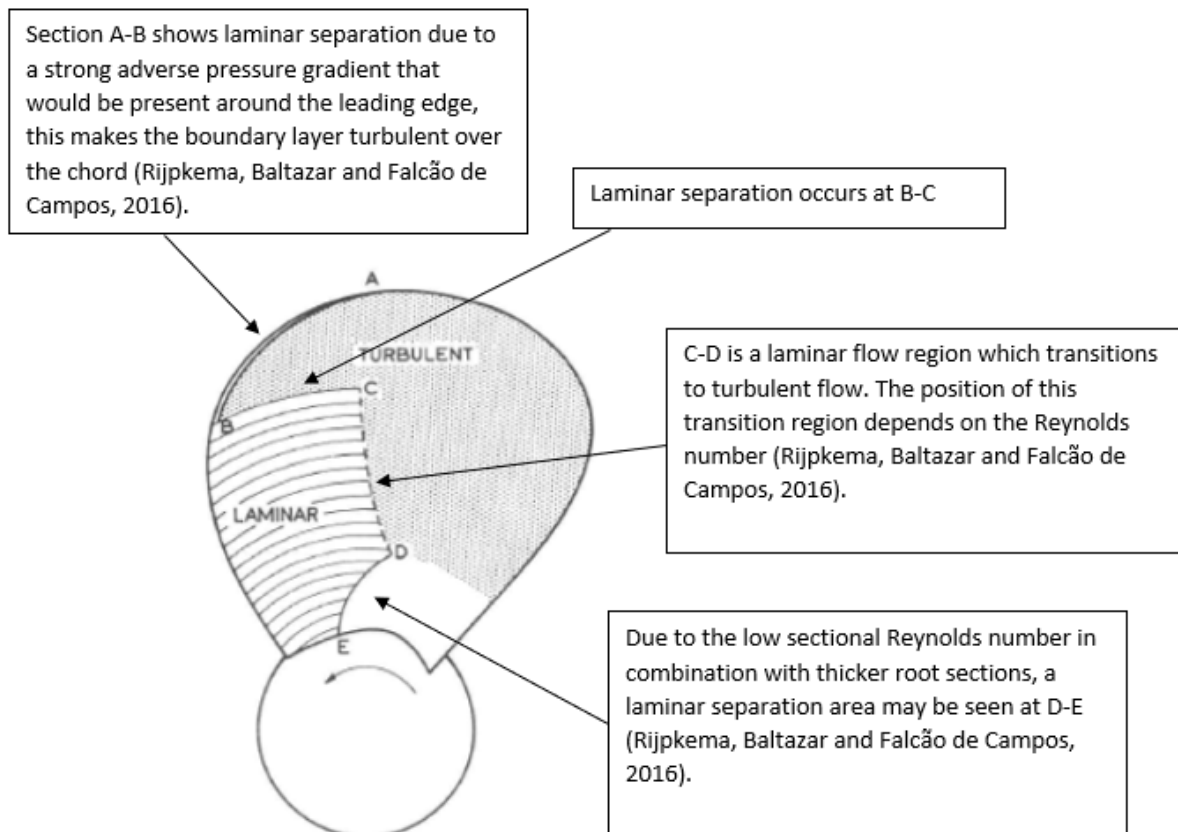


Figure 3- Representation of boundary layer on suction side of propeller (Rijpkema, Baltazar and Falcão de Campos, 2016, p.2).

The flow over the propeller at model-scale Reynolds numbers is mostly in the critical Reynolds number regime, where laminar, transition and turbulent flow regimes may occur simultaneously (Rijpkema, Baltazar and Falcão de Campos, 2016).

3.2 How Reynolds Number affects the Boundary Layer

Variations in Reynolds number have a significant impact on propeller performance and boundary-layer flow.

For lower Reynolds numbers, the radial direction of the boundary-layer flow due to centrifugal force is seen below. In comparison to higher Reynolds numbers, the increased radial component results in a distinctly different flow pattern due to the larger shear-stress component. The limiting streamlines become more circumferentially oriented as Reynolds number rises (Rijpkema, Baltazar and Falcão de Campos, 2016).



With increasing Reynolds number, the thickness of the boundary layer visually reduces, with the $Re = 10^4$ result displaying a thicker boundary layer than the results at the higher Reynolds numbers.

The boundary layer on the suction side is thicker than on the pressure side, indicating that the suction side's adverse pressure gradient is slowing the flow (Rijpkema, Baltazar and Falcão de Campos, 2016).

Figure 4: "Contour line of total pressure loss ($C_{pl} = 0.01$) representing the boundary-layer shape for $J=0.7$ for E779A propeller at $0.7R$ " (Rijpkema, Baltazar and Falcão de Campos, 2016 p.8)

3.2 How Reynolds Number affects the Boundary Layer

An investigation of boundary-layer flow reveals that lower Reynolds numbers have more radially directed flow, while higher Reynolds numbers have more circumferentially directed flow. With rising Reynolds numbers, the separation regions at both the leading and trailing edges diminish (Rijpkema, Baltazar and Falcão de Campos, 2016)

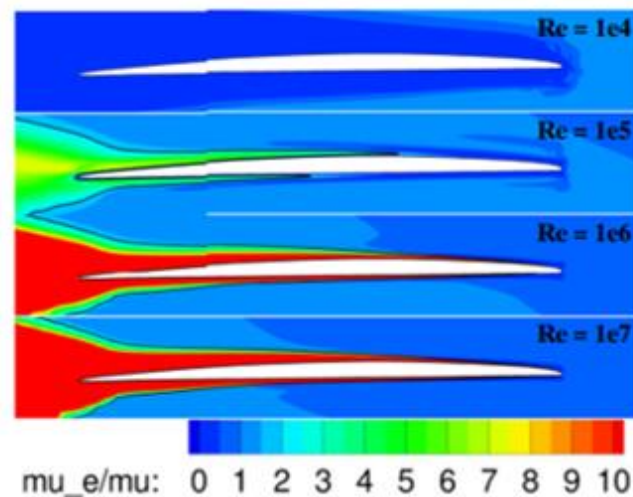


Figure 5: Contour plot of eddy-viscosity ratio (contour line at $\mu_e/\mu = 2$) for $J=0.7$ for E779A propeller at $0.7R$ (Rijpkema, Baltazar and Falcão de Campos, 2016, p.10)

A simple depiction of the boundary-layer form shown above reveals a thicker boundary layer for lower Reynolds numbers, which impacts the pressure distribution. The eddy viscosity distribution shows that turbulence is inactive at lower Reynolds numbers and becomes more active as Reynolds numbers rise; this is generally the same for the majority of propeller designs where the turbulence around the blade occurs at the trailing edge (Rijpkema, Baltazar and Falcão de Campos, 2016).

The ratio of Turbulent Eddy Viscosity to Dynamic Viscosity is known as the Eddy Viscosity Ratio. It is utilised in CFD simulations to account for the lost effects when the turbulent effects are averaged. It simulates the energy transmission and dissipation, which was neglected due to turbulence modelling (SimScale, 2019).

3.2 How Reynolds Number affects the Boundary Layer

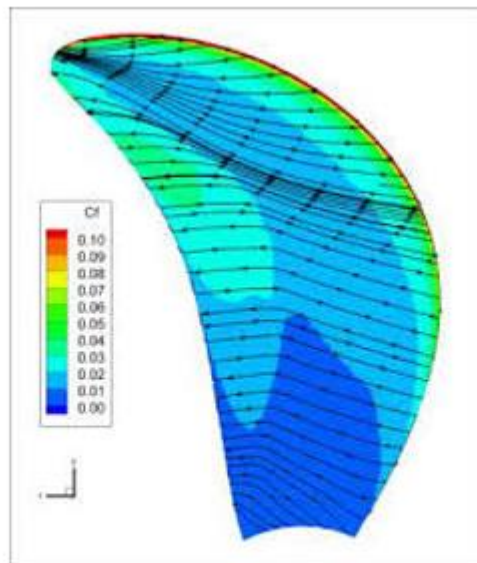


Figure 6: Diagram showing turbulence present at the trailing edge (as described on the previous page) (Li, Lindell and Werner, 2019)

3.3 How Propeller Design affects Performance

Curved blades slice through the water (leading edge) and produce lift to propel the ship ahead. The blade face is under a lot of pressure and the blade back is at a lower pressure. The curvature of the blade is increased to delay separation, reduce the amount of turbulence around the ship's propeller and generate more power (Aerodynamics4students, 2019).

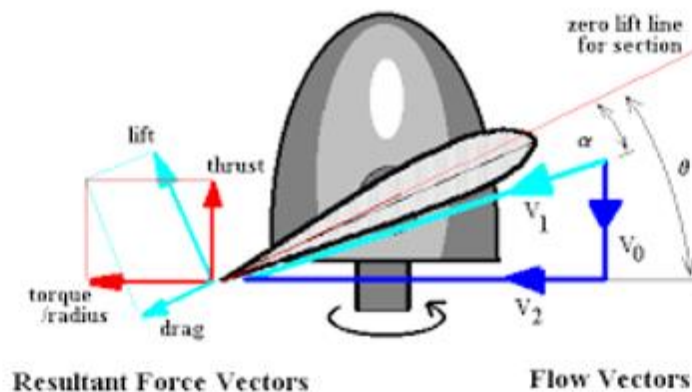


Figure 7: Image depicting how local angle of attack affects vectors (Aerodynamics4students, 2019)

The local angle of attack is an essential feature of the propeller blade. It is reliant on the blade's leading edge cutting through the water. The local angle of attack determines the amount of lift generated. As the propeller spins faster, the local angle of attack increases, resulting in increased lift; therefore, the blades must be properly pitched. The flow will begin to separate as the angle of attack increases, and high speeds will cause cavitation to occur (Colley, 2012). Too high of an angle of attack will cause a loss of lift known as stall.

Skewing a ship propeller greatly reduces the amount of unstable hydrodynamic loading (wake). Because of the viscous 'drag' effects, this has an indirect positive effect in reducing resistance (Ghosh, 2020).

The propeller blade, in this case, is designed (long length blades) to give a large Reynolds number to allow the turbulent region to be at the trailing edge of the propeller. However, too great of a diameter would lead to high shaft torque (Rijpkema, Baltazar and Falcão de Campos, 2016).

Increasing the Reynolds number will increase thrust and reduce torque, which results in improved open-water efficiency (Rijpkema, Baltazar and Falcão de Campos, 2016).

Optimising the design for cost is very important; the more complex and challenging to develop the design, the larger the costs; therefore, a compromise between cost and performance needs to be established.

4.0 Task 3

This report section will briefly cover why high tensile brass has been superseded by NAB and compare CFRP to NAB and GFRP.

4.1 CFRPs Implications on Propeller Performance

Carbon fibre has a low weight to high strength ratio, which allows it to be useful in a wide range of engineering applications. Many researchers have been conducting studies on the use of CFRP in marine propellers conducting experiments to investigate the material's effect on performance and trying to develop the optimum stacking sequence. A general consensus from research is that many researchers are comfortable with using CFRP as an alternative material to the traditionally used metals (Islam et al., 2022).

CFRP can increase propulsion performance by decreasing vacuum waves (cavitation) created during navigation (tanchain, 2022).

A study was performed on the effect of the stacking sequence of CFRP propellers. The outcome found that using an optimal stacking sequence and ply angle for CFRP propellers reduces hub loads by 50-70% and, therefore, improves the ship's fuel efficiency (He, Hong and Wang, 2012).

A study on heavy loading conditions for long stroke slow-speed engines showed that existing propellers are not suitable for reaching the high loading requirements that CFRP can reach (Lee et al., 2017).

CFRP can be designed for specific physical, mechanical, or acoustic properties giving much more flexibility than traditional metals (Uddin, Hossen, Jahan and Islam, 2021).

During a study, it was found that CFRP composites are sensitive to the seawater environment. The results from the study showed a slight decrease in fatigue lifetime under tension from water ingress. However, a much more significant decrease was present when the sample was under four-point flexural fatigue (Le Guen-Geffroy et al., 2018).

Studies have discussed possibilities to reduce water ingress, such as the introduction of a nanoparticle (eg. clay). To attain the desired compatibility, a functionalising agent such as silane may be required (Kamra et al., 2015).

Efforts have also been made to improve the flexibility of design for composites to improve their performance (Das and Kapuria, 2016). These performance advantages can take the form of reduced dynamic stress (He, Hong and Wang, 2012), increased adaptivity to spatially variable wake (Mulcahy, Prusty and Gardiner, 2010), and increased hydrodynamic efficiency (Young et al., 2016).

4.2 High Tensile Brass

High Tensile Brass was historically the most popular material for ship propellers in the mid-1900s. They were easy to melt and cast and had high strength. The main issue with this material was corrosion resistance; the addition of aluminium overcame this. If tin is omitted from the material, the brass alloy will corrode rapidly; high tin content, however, will lead to the propeller becoming more sensitive to stress corrosion cracking (Carlton, 2019).

High Tensile Brass was replaced in favour of NAB due to NAB possessing much greater resistance to corrosion and higher yield strength.

4.3 Discussion and Comparison of CFRP and NAB

Today, propellers in ships and submarines are usually made of nickel-aluminium-bronze (NAB) alloy due to the material's excellent corrosion resistance and high yield strength. However, these propellers are susceptible to a range of issues such as its high cost, corrosion fatigue, vibration and significant weight. However, CFRP offers advantages to overcome these issues, such as greater damping to reduce vibrations, much lighter weight, and less need for a corrosion protection layer (Islam et al., 2022).

CFRP's other advantages compared to NAB include a greater strength to weight ratio, reduced maintenance costs, higher corrosive resistance, and better fatigue performance (Wozniak, 2005).

Recently, more research is being conducted to overcome CFRP's two main disadvantages: cavitation erosion and resistance to seawater environment.

In a study comparing the vibration and cavitation erosion resistance of composites and NAB for ship propellers, it was found that the damping ratio of CFRP was ten times greater than NAB. Cavitation, however, was a large problem for CFRP. NAB's cavitation erosion resistance was far greater than CFRP's. Cavitation erosion will lead to the eventual failure of the propeller. The use of Aramid Fibre is in discussion to improve erosion resistance (Yamatogi et al., 2011).

A structural analysis study was performed in 2017 comparing CFRP and NAB; the results concluded that CFRP produced more deflection in radial and chordal directions than NAB (Rubino, Nisticò, Tucci and Carlone, 2020). This is seen as an advantage; from research, it has been deduced that flexible composite propellers offer more potential for improved performance over metals concerning vibration and cavitation. The blade will deform due to hydrodynamic loading, and this has the effect of reducing hull vibration and cavitation. Two propellers (one rigid and one flexible) were compared in a study. During testing, it was found that the cavitation performance was better in the flexible propeller (Chen et al., 2006).

The limitations of NAB have pushed engineers to look for alternative materials for the production of propeller blades, with the most notable research being performed on FRP and glass epoxy composites (Rubino, Nisticò, Tucci and Carlone, 2020).

4.4 Discussion and Comparison of CFRP and GFRP

Glass fibre composites are being discussed by researchers for use in propellers in the future due to their great potential for energy absorption and ability to design to improve specific characteristics.

The consensus is that glass fibre composites are more impact resistant than carbon fibre composites. Despite having higher strength and modulus than most technical fibres, carbon fibre composites have a lower impact resistance than glass fibre composites due to their brittle nature. This is due to carbon fibres' lower strain-to-failure, which ranges from 0.5 to 2.5 percent (Islam et al., 2022).

However, carbon fibre typically outperforms glass fibre in terms of strength, stiffness, and moisture absorption (Islam et al., 2022).

During a study of erosion resistance, it was found that CFRP had greater erosion resistance than GFRP, but in comparison to NAB both are poor (Yamatogi et al., 2009).

In 2020 it was found that no studies had been performed to compare CFRP and GFRP composites that have the same matrix material through indentation tests. Glass composites were found to absorb more energy than carbon composites. The results showed that GFRP had almost twice the total energy absorption of CFRP (Fotouhi et al., 2020).

Water absorption is a persistent issue with fibre composites. For GFRP water absorption could be reduced by reducing micro-void creation, which could be researched further (Dhakal, MacMullen and Zhang, 2016).

A study comparing CFRP, GFRP and NAB showed that CFRP had the least stress/strain generated at various loads. CFRP has better structural properties whilst having the lowest mass (40% lighter than GFRP and 80% lighter than NAB in this study). GFRP's structural properties were marginally below NAB's whilst having a much lower mass. The composites are stronger, lighter and have better corrosion and chemical resistance than traditional metal propellers. Therefore, composites are being employed to overcome the disadvantages of metal propellers (Uddin, Hossen, Jahan and Islam, 2021).

4.5 Relevant Industry Standards and Codes of Practice

Relevant Codes of Practice	Description
Code of safe working practices for merchant seafarers (COSWP) amendment 6 2021	According to codes of practice “a United Kingdom fishing vessel must not proceed on a voyage unless— (a) the Secretary of State has issued a certificate of compliance in respect of that vessel; (b) that certificate of compliance is valid at the time of the voyage; and (c) the vessel continues to comply with the applicable Code of Practice” (Gov.UK, 2017). These codes also includes a list of approved materials, carbon fibre is not on the list of approved materials in the UK and must be authorised by the load line assigning authority.
MERCHANT SHIPPING The Fishing Vessels (Codes of Practice) Regulations 2017	

Table 1 – Relevant Codes of Practice

Relevant Industry Standards	Description
ISO 19030-1:2016	It defines a set of performance indicators for hull and propeller maintenance, repair, and retrofit activities, as well as general concepts for measuring changes in hull and propeller performance.
ISO 3715-2:2001	List the requirements for controlled pitched propellers (CFRP for example) to be approved.
ISO 20233-1:2018 ISO 20233-2:2019	Are to be followed when performing testing related to cavitation performance.

Table 2 – Relevant Industry Standards

5.0 Task 4

A wide range of models are used for CFD simulations that cover different parts of fluid mechanics. Each modelling technique will have its unique benefits and problems, but this section will be brief and cover the advantages and disadvantages of CFD, FEA, FVM and FDM (FEA, FVM and FDM are all meshes used in CFD).

5.1 Advantages and Disadvantages of CFD

Computers are becoming more powerful as time goes on, allowing for more demanding mathematical methods of numerical computation. FVM (Finite-Volume Method) is one such method, which is used to discretize the Navier-Stokes differential equations, which regulate the behaviour of fluids in nature. This method also serves as a foundation for CFD (Computational Fluid Dynamics) analysis, a discipline of fluid mechanics that employs numerical analysis to address issues involving fluid flows (Sola, García-Baena and Jesús, 2016).

Advantages of CFD	Disadvantages of CFD
Using CFD to predict the functioning of design elements at an early stage reduces the risks of any physical failures occurring during application. Reducing the risks reduces the likelihood of design changes in the later stages of a project which can be very costly. Uncertainties are defined sooner, allowing risks to be designed out when it is much simpler. This is the main justification for using CFD and other simulation/modelling software.	The behaviour of turbulence is not fully understood. Therefore it can only be modelled with a large amount of approximation involved. Due to this, there are many potential sources of errors because of approximations used; therefore, it is essential to verify and validate simulations.
Performing a CFD simulation is inexpensive, and with the use of more powerful PC's, the costs are likely to decrease further (Pretechnologies, 2021).	A lot of computational power and time is needed to perform complex simulations.
CFD allows examination of locations of interest in a model and interpretation of the performance in this region by changing flow parameters (Pretechnologies, 2021).	CFDs are prone to conversion issues depending on the quality of the mesh. To reduce conversion issues, a high-quality mesh will need to be used.
Any physical condition can be theoretically simulated with CFD (Pretechnologies, 2021).	Simple flow models or simplified boundary conditions may lead to errors (ifes, 2020).
CFD gives control over the physical process and allows the isolation of individual phenomena for research (Pretechnologies, 2021).	Potential uncertainties stemming from insufficient computing values can result in interpolation errors (ifes, 2020).
CFD allows the study of a system under conditions well beyond its limits (CFD Modeling Services, 2013).	
The level of detail that CFD provides is fit for any need and can be further improved when using better equipment.	
A wide range of flow parameters can be gathered, even ones that cannot be obtained through experimentation.	

Table 3 - Advantages and Disadvantages of CFD

5.2 Advantages and Disadvantages of FEA/FEM

The finite-element approach divides a CAD model into very small yet finite-sized geometrically simple elements. The finite-element mesh is made up of all of these simple shapes (Sjodin, 2016).

The approach was first used for structural analysis in the past. The finite element method has been discovered to be suitable for a large class of multi-physics issues such as fluid flow (Sjodin, 2016).

Advantages of FEA/FEM	Disadvantages of FEA/FEM
Comprehensive result sets can be obtained, generating the system's physical response at any location, including some that could have been overlooked in a traditional analytical approach (Pretechnologies, 2022). Simulation of dangerous load conditions and failure modes can be performed safely (Pretechnologies, 2022). Several failure modes or physical events can be tested within the same model (Pretechnologies, 2022). It takes the same amount of effort to model any shaped geometry (autodesk, 2019). Costs are usually much lower when compared to physical experiments.	FEA takes a long time to solve large problems and is not well suited for turbulent flow. Standard FEA cannot be utilised to solve every CFD problem without modification due to instabilities (Cadence CFD, 2021). Navier-Stokes equations require a more numerical approach to be solved easily and so FEM is not a good choice as it is challenging to solve hyperbolic equations by using FEM. Using FVM would be an excellent solution (Cadence CFD, 2021).

Table 4 – Advantages and Disadvantages of FEA/FEM

5.3 Advantages and Disadvantages of FVM

One of CFD's most adaptable discretization techniques is the Finite Volume Method (FVM) (Sjodin, 2016).

Like the finite element method, the finite volume approach is based on an unstructured mesh. As a result, it works well with irregular and complex geometry. FVM provides a large benefit over FEM for fluid problems because it is based on the integral form of the conservation laws rather than their differential form. This results in increased accuracy and stability, particularly for sharp gradients (i.e. big derivatives) inside a domain (Neill and Hashemi, 2018).

Advantages of FVM	Disadvantages of FVM
Finite volume control does not limit cell shape; it also conserves mass, momentum and energy. Non-conservative schemes are often less accurate than conservative schemes, especially in the presence of large gradients (Kolditz, 2002). To approximate complex geometries, finite volume techniques take full advantage of arbitrary meshes (Kolditz, 2002). FVM is considered easier to program than FEM (Sjodin, 2016) FVM uses an unstructured mesh which offers more flexibility for working with complex geometries. Costs are usually much lower when compared to physical experiments.	False diffusion (and error having a diffusion-like appearance) can occur when simple numerics are used. Higher than second-order methods are more challenging to develop in 3D using FVM but is much easier to perform using FDM (Sjodin, 2016).

Table 5 – Advantages and disadvantages of FVM

5.4 Advantages and Disadvantages of FDM

The most straightforward way for discretizing partial differential equations is the finite-difference method. The finite-difference method is usually defined on a regular grid and is useful for solving problems quickly. As a result, the approach is more commonly employed for rectangular or block-shaped models than irregular CAD geometries (Sjodin, 2016).

Advantages of FDM	Disadvantages of FDM
Considered the easiest to implement out of the discussed modelling techniques (Sjodin, 2016). Easier to obtain higher-order schemes than other mesh types on regular grids (Sjodin, 2016). Costs are usually much lower when compared to physical experiments.	It is restricted to simple geometries, so it will not be as helpful as FVM in simulating fluid flow. Does not conserve momentum, energy, and mass on coarse grids.

Table 6 – Advantages and disadvantages of FDM

Note: Due to their reliance on the mesh topology, multiphase flows, free-surface flows, and shifting geometries can be difficult to simulate when using FDM, FVM and FEM, but they can all be used to simulate fluid flows accurately (Sabrowski and Tschirschnitz, 2020).

6.0 Task 5

6.1 Experiment Introduction

This section will discuss an experiment to validate a CFD simulation.

Validation is evaluating a computer simulation's accuracy by comparing it to experimental data, and it is one of the most used approaches for building confidence in simulation results.

There are many methods of experimentally validating CFD simulations of fluid flow around a ship propeller. In this section, Particle image velocimetry (PIV) will be considered to understand the fluid flow around a propeller prototype. This validation method is an effective tool for understanding the water flow around a propeller.

Computational fluid dynamics will be used to produce a simulation of the fluid flow around the proposed prototype model. CFD is a cheap and effective tool that can numerically study fluids' flow characteristics.

This experiment will be performed in three steps.

Step 1: Producing the computational model and mesh based on the physical features of the prototype.

Step 2: Calculate the fluid flow solution.

Step 3: Analysing the results - This will include validating the CFD model by comparing the simulation result to the experimental result.

In this experiment, a CFD model will be compared to a PIV experiment of the prototype propeller.

As the main focus of this task will be a discussion as to how the physical experiment and acquisition of data will be performed, the CFD modelling will be discussed in appendix A.

6.2 PIV Experiment Set-up and Testing

After processing and analysing the CFD simulation, the physical experiment for comparison will be performed.

The PIV experimental study to evaluate the CFD models/simulation will be set up as shown below.

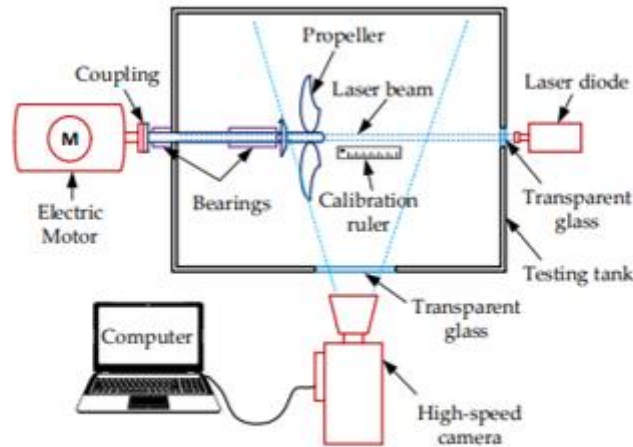


Figure 8: PIV Experiment Set-up (Overhead View) (Nyongesa et al., 2022)

The testing tank will be filled with water that matches the values used for viscosity and density in the CFD simulation to give the same Reynolds number.

The high-speed camera will estimate the flow characteristics. The laser diode is used to illuminate the seeding particles so the high-speed camera can capture the fluid flow around the propeller.

Photron Fastcom software will be used for the camera, the camera will be set at 250 frames per second and the images will be adopted to produce the velocity distribution of the flow fields. In this case, lower RPMs will be used, but it is noted that at high RPM values (200+), bubble formation will interfere with the laser and reduce measurement accuracy (Nyongesa et al., 2022).

Before testing, the camera will be calibrated and the calibration image will be used to convert a frame's velocity information from frame to velocity (m/s) during the analysis of results (Nyongesa et al., 2022).

PIVlab software is used to analyse PIV experiments by evaluating the time-resolved PIV images acquired by the high-speed camera (Nyongesa et al., 2022).

6.2 PIV Experiment Set-up and Testing

Data validation is a feature that can be used in post-processing. Outlier points will be removed from the PIV data. PIVlab allows the selection of acceptable velocity limits and filters out the outliers (Nyongesa et al., 2022).

Noise is a general issue in PIV testing, but it can be reduced by applying data smoothing through PIVlab (Nyongesa et al., 2022).

One disadvantage of this test method is that during data collection, image quality can vary due to distortion of the flow field; this can produce results with some errors during low-quality data collection. To overcome this issue, glass hollow spheres will be used as seeding particles to ensure the resolution of the scattering images is of good quality without any distortion of the flow field.

This solution of using glass hollow spheres was provided in a documented PIV experiment and showed that the visualisation of the flow distribution was significantly improved (Meng et al., 2019).

Another disadvantage would be that luminance noise can cause incorrect readings for turbulence levels. A laser line filter is used on the high-speed camera to reduce luminance noise from other wavelengths other than the scattering signal (Nyongesa et al., 2022).

6.3 Examples of PIV Experiment Results

The following are figures showing examples of results of this type of experiment being compared with a CFD model. Data acquisition and plotting are all performed through settings within PIVlab software.

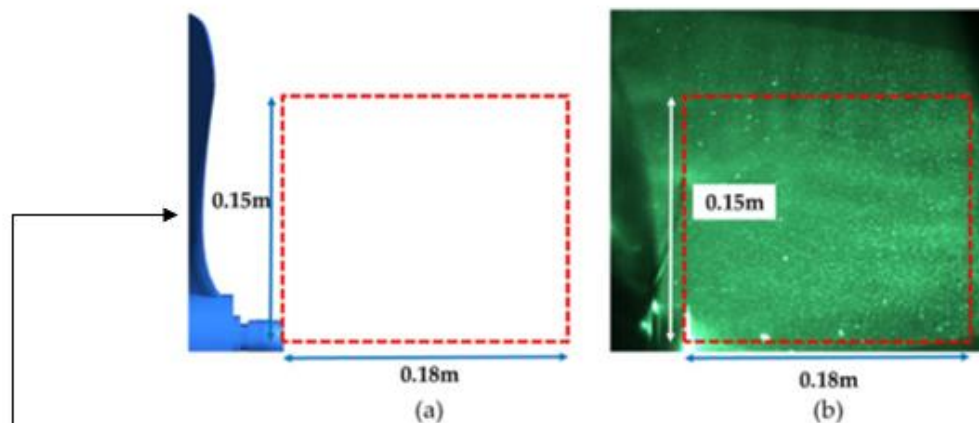


Figure 9: Typical region of interest size for CFD (a) and PIV experiment (b) (Nyongesa et al., 2022, p.12)

A region of interest is selected and velocity vectors, velocity magnitude colour maps, and velocity magnitude values are produced through the data acquired to analyse the entire water flow characteristics behind the propeller. This is then compared to the CFD model, therefore, comparing the flow characteristics behind the propeller (Nyongesa et al., 2022).

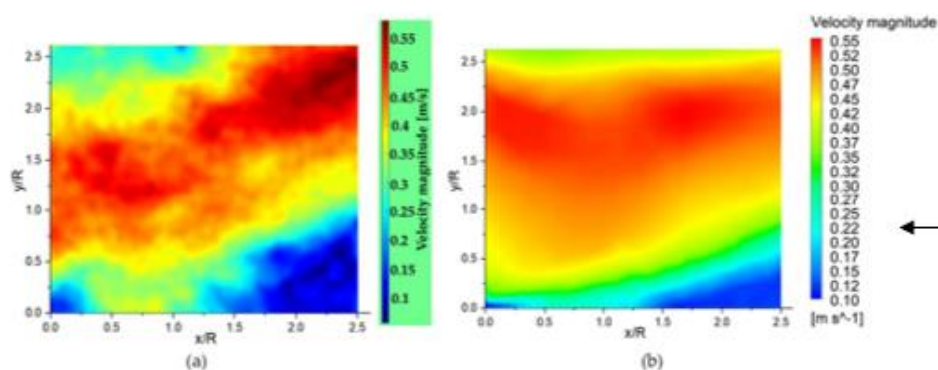


Figure 10: Velocity Magnitude colour Map for CFD (a) and PIV experiment (b) (Nyongesa et al., 2022, p.12)

6.3 Examples of PIV Experiment Results

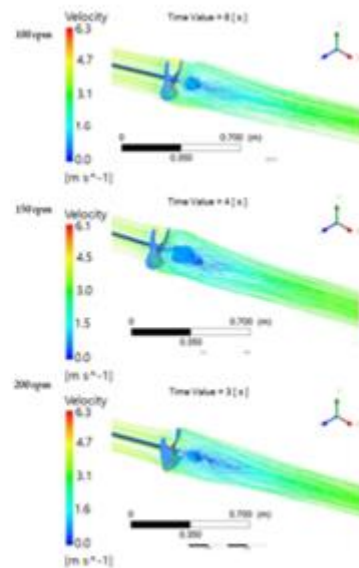


Figure 11: Example of Velocity Streamline Result (Nyongesa et al., 2022, p.16)

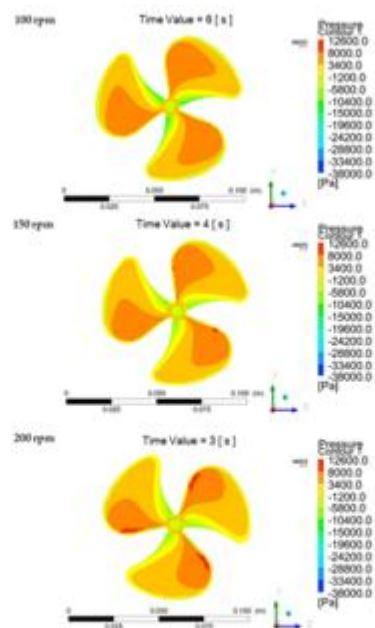


Figure 12: Pressure distribution on the

Figure 12: Example of Propeller Pressure Distribution Result (Nyongesa et al., 2022, p.18)

Deviations can be attained between the CFD model and the physical experiment by comparing the results. If the deviation is low (less than 5%), the CFD model produced is suitable for modelling the fluid flow around the propeller.

6.4 Cost Saving Method

Producing a propeller to the scale detailed within task one would significantly increase the cost of producing the propeller and the cost of the PIV test set-up. To reduce these costs and save space, using a scaled model can be employed.

In this case, a 1/10 scale model would be of suitable size for fitting into an adequately sized tank. However, scaled-down models will give different results than the full-scale model and a new CFD simulation would need to be carried out.

When designing a scaled-down model, there are two important values to consider: Reynolds number and the displacement ratio (refer to Appendix B for additional information).

Fluid flow is very complex, and there will inevitably be differences between the scaled-down and real-life models. A comparison study can potentially be carried out between two scaled-down models (1/10 and 2/10) to verify that the model will perform accurately to the full-scale propeller when considering Reynolds number and displacement ratio. A comparison between the scaled-down CFD simulations and calculations would also need to be performed.

7.0 Conclusion

It was found that Reynolds number has a large influence on the flow around a propeller blade. Larger Reynolds numbers delay the separation point and improves the ship's open water efficiency.

Propeller design contributes massively to a vessel's performance, and much research has been performed on how to optimise the design of varying materials to give the best performance whilst complying with relevant industry standards and codes of practice.

It was found that carbon fibre had many advantages over the current most popular material for ship propellers (NAB). Much research has gone into eliminating its weaknesses, including cavitation erosion and resistance to water ingress.

It was found that different modelling techniques for CFD are more suitable than others for specific activities, such as FVM being the best meshing method compared to FDM and FEA for fluid flow analysis.

PIV experimentation is suitable for the simulation of flow when considering and overcoming its disadvantages through the use of glass hollow balls to reduce image distortion and a laser line filter to reduce luminance noise.

Using scaled models for verification and validation is a viable method for heavily reducing the cost of testing whilst giving accurate results relative to the full-scale model when considering the displacement ratio and Reynolds number.

8.0 Works Cited

- Aerodynamics4students, 2019. *Blade Element Propeller Theory | Aerodynamics for Students*. [online] Aerodynamics4students.com. Available at: <<http://www.aerodynamics4students.com/propulsion/blade-element-propeller-theory.php>> [Accessed 4 June 2022].
- autodesk, 2019. *Finite Element vs Finite Volume | CFD | Autodesk Knowledge Network*. [online] Knowledge.autodesk.com. Available at: <<https://knowledge.autodesk.com/support/cfd/learn-explore/caas/CloudHelp/cloudhelp/2014/ENU/SimCFD/files/GUID-12A9AED8-2047-4D3A-BC80-82BE9CF47517-htm.html>> [Accessed 5 June 2022].
- Cadence CFD, 2021. *FEA vs. CFD: The Differences and Applications of Simulation Tools*. [online] Cadence System Analysis. Available at: <<https://resources.system-analysis.cadence.com/blog/msa2021-fea-vs-cfd-the-differences-and-applications-of-simulation-tools>> [Accessed 5 June 2022].
- Cadence CFD, 2022. *What Is Reynolds Number and Why Is It Important?*. [online] Cadence System Analysis. Available at: <<https://resources.system-analysis.cadence.com/blog/msa2022-what-is-reynolds-number-and-why-is-it-important>> [Accessed 4 June 2022].
- Carlton, J., 2019. *Marine Propellers and Propulsion*. 4th ed. London: Butterworth-Heinemann, pp.425-436.
- CFD Modeling Services, 2013. *4 Advantages and Disadvantages of CFD*. [online] CFD Modeling Services. Available at: <<https://cfdmodelingservices.wordpress.com/2013/06/12/4-advantages-and-disadvantages-of-cfd/>> [Accessed 5 June 2022].
- Chen, B., Neely, S., Michael, T., Gowing, S., Szwer, R., Buchler, D. and Schult, R., 2006. Design, Fabrication and Testing of the Composite Pitch-Adapting (Flex) Propellers. *Paper presented at the SNAME 11th Propeller and Shafting Symposium*, [online] 1(1), p.1. Available at: <<https://onepetro.org/snamepss/proceedings-abstract/PSS06/1-PSS06/D011S001R008/477654>> [Accessed 4 June 2022].
- Colley, E., 2012. *Analysis of Flow around a Ship Propeller using OpenFOAM*. [online] Cmst.curtin.edu.au. Available at: <https://cmst.curtin.edu.au/wp-content/uploads/sites/4/2016/05/colley_eamonn_2012_analysis_of_flow.pdf> [Accessed 4 June 2022].
- Das, H. and Kapuria, S., 2016. On the use of bend-twist coupling in full-scale composite marine propellers for improving hydrodynamic performance. *Journal of Fluids and Structures*, [online] 61, pp.132-153. Available at: <<https://www.sciencedirect.com/science/article/abs/pii/S0889974615002625>> [Accessed 4 June 2022].

8.0 Works Cited

- Dhakal, H., MacMullen, J. and Zhang, Z., 2016. Moisture measurement and effects on properties of marine composites. *Marine Applications of Advanced Fibre-Reinforced Composites*, [online] pp.103-124. Available at: <https://www.researchgate.net/publication/301253603_Moisture_measurement_and_effects_on_properties_of_marine_composites> [Accessed 4 June 2022].
- Drózdź, A., Niegodajew, P., Romańczyk, M. and Elsner, W., 2021. Effect of Reynolds number on turbulent boundary layer approaching separation. *Experimental Thermal and Fluid Science*, [online] 125(1), p.110377. Available at: <<https://www.sciencedirect.com/science/article/pii/S0894177721000297>> [Accessed 4 June 2022].
- Fotouhi, M., Damghani, M., Leong, M., Fotouhi, S., Jalalvand, M. and Wisnom, M., 2020. A comparative study on glass and carbon fibre reinforced laminated composites in scaled quasi-static indentation tests. *Composite Structures*, [online] 245(1), pp.1-14. Available at: <<https://www.sciencedirect.com/science/article/pii/S0263822320313647>> [Accessed 4 June 2022].
- Ghosh, S., 2020. *Understanding Design Of Ship Propeller*. [online] Marineinsight.com. Available at: <<https://www.marineinsight.com/naval-architecture/design-of-ship-propeller/>> [Accessed 4 June 2022].
- Gov.UK, 2017. *MERCHANT SHIPPING The Fishing Vessels (Codes of Practice) Regulations 2017*. [online] Legislation.gov.uk. Available at: <https://www.legislation.gov.uk/ukxi/2017/943/pdfs/ukxi_20170943_en.pdf> [Accessed 5 June 2022].
- Hara, Y., Yamatogi, T., Murayama, H., Uzawa, K. and Kageyama, K., 2015. *PERFORMRANCE EVALUATION OF COMPOSITE MARINE PROPELLER FOR A FISHING BOAT BY FLUID-STRUCTURE INTERACTION ANALYSIS*. [online] Iccm-central.org. Available at: <[https://www.iccm-central.org/Proceedings/ICCM18proceedings/data/2.%20Oral%20Presentation/Aug22\(Monday\)/M27%20Applications%20of%20Composites/M27-6-IF1726.pdf](https://www.iccm-central.org/Proceedings/ICCM18proceedings/data/2.%20Oral%20Presentation/Aug22(Monday)/M27%20Applications%20of%20Composites/M27-6-IF1726.pdf)> [Accessed 5 June 2022].
- He, X., Hong, Y. and Wang, R., 2012. Hydroelastic optimisation of a composite marine propeller in a non-uniform wake. *Ocean Engineering*, [online] 39(1), pp.14-23. Available at: <<https://www.sciencedirect.com/science/article/pii/S0029801811002381>> [Accessed 4 June 2022].
- He, X., Hong, Y. and Wang, R., 2012. Hydroelastic optimisation of a composite marine propeller in a non-uniform wake. *Ocean Engineering*, [online] 39, pp.14-23. Available at: <<https://www.sciencedirect.com/science/article/abs/pii/S0029801811002381>> [Accessed 4 June 2022].
- ifes, 2020. *Computational Fluid Dynamics (CFD)*. [online] Ifes-koeln.de. Available at: <<https://www.ifes-koeln.de/en/services-20-years-project-experience/simulation-analysis/computational-fluid-dynamics-cfd.html>> [Accessed 4 June 2022].

8.0 Works Cited

- Islam, F., Caldwell, R., Phillips, A., John, N. and Prusty, B., 2022. A review of relevant impact behaviour for improved durability of marine composite propellers. *Composites Part C: Open Access*, [online] 8(1), pp.1-22. Available at: <<https://www.sciencedirect.com/science/article/pii/S2666682022000202>> [Accessed 4 June 2022].
- Kamra, T., Chaudhary, S., Xu, C., Johansson, N., Montelius, L., Schnadt, J. and Ye, L., 2015. Covalent immobilization of molecularly imprinted polymer nanoparticles using an epoxy silane. *Journal of Colloid and Interface Science*, [online] 445(1), pp.277-284. Available at: <<https://www.sciencedirect.com/science/article/pii/S002197971401056X>> [Accessed 4 June 2022].
- Kolditz, O., 2002. Finite Volume Method. *Computational Methods in Environmental Fluid Mechanics*, pp.173-190.
- Li, D., Lindell, P. and Werner, S., 2019. Transitional Flow on Model Propellers and Their Influence on Relative Rotative Efficiency. *Journal of Marine Science and Engineering*, 7(12), p.12.
- Le Guen-Geffroy, A., Le Gac, P., Diakhate, M., Habert, B. and Davies, P., 2018. Long-term durability of CFRP under fatigue loading for marine applications. *MATEC Web of Conferences*, [online] 165(1), p.7. Available at: <<https://archimer.ifremer.fr/doc/00441/55262/56756.pdf>> [Accessed 4 June 2022].
- Lee, H., Hur, J., Park, J., Hah, Z., Song, M. and Chang, B., 2017. A Design Concept of Composite Marine Propeller for Long Stroke Slow Speed Main Engine. *Fifth International Symposium on Marine Propulsors*, [online] 1(1), pp.1-7. Available at: <<https://www.marinepropulsors.com/proceedings/2017/WA2-1.pdf>> [Accessed 4 June 2022].
- Meng, S., Li, X., Yan, X., Wang, L., Zhang, H. and Cao, Y., 2019. Turbulence Models for Single Phase Flow Simulation of Cyclonic Flotation Columns. *Minerals*, [online] 9(8), p.464. Available at: <<https://www.mdpi.com/2075-163X/9/8/464/htm>> [Accessed 5 June 2022].
- MIT News Office, 2020. *Understanding how fluids heat or cool surfaces*. [online] MIT News | Massachusetts Institute of Technology. Available at: <<https://news.mit.edu/2020/how-fluids-heat-cool-surfaces-0428>> [Accessed 5 June 2022].
- Mulcahy, N., Prusty, B. and Gardiner, C., 2010. Hydroelastic tailoring of flexible composite propellers. *Ships and Offshore Structures*, [online] 5(4), pp.359-370. Available at: <<https://www.tandfonline.com/doi/abs/10.1080/17445302.2010.481139>> [Accessed 4 June 2022].
- Neill, S. and Hashemi, M., 2018. Ocean Modelling for Resource Characterization. *Fundamentals of Ocean Renewable Energy*, [online] pp.193-235. Available at: <<https://www.sciencedirect.com/science/article/pii/B9780128104484000082>> [Accessed 5 June 2022].
- Nyongesa, A., Pham, V., Yoon, S., Kwon, W., Kim, J., Ngo, D., Choi, J., Sul, Y. and Lee, W., 2022. Investigation of the Effect of Rope Cutter on Water Flow behind Ship Propellers Based on CFD Analysis. *Machines*, [online] 10(5), pp.1-27. Available at: <<https://www.mdpi.com/2075-1702/10/5/300>> [Accessed 5 June 2022].

8.0 Works Cited

Pretechnologies, 2021. *Advantages of CFD | FEA services and consultancy. CFD and FEA company.*

[online] Pretechnologies.com. Available at:

<<http://www.pretechnologies.com/services/computational-fluid-dynamics/advantage>> [Accessed 4 June 2022].

Pretechnologies, 2022. *Advantages of FEA | FEA services and consultancy. CFD and FEA company.*

[online] Pretechnologies. Available at: <<http://www.pretechnologies.com/services/finite-element-analysis/advantages>> [Accessed 5 June 2022].

Rijpkema, D., Baltazar, J. and Falcão de Campos, J., 2016. *Viscous Flow Simulations of Propellers in Different Reynolds Number Regimes.* [online] ResearchGate. Available at:

<https://www.researchgate.net/publication/278703552_Viscous_Flow_Simulations_of_Propellers_in_Different_Reynolds_Number_Regimes> [Accessed 4 June 2022].

Rubino, F., Nisticò, A., Tucci, F. and Carlone, P., 2020. Marine Application of Fiber Reinforced

Composites: A Review. *Journal of Marine Science and Engineering*, [online] 8(1), p.26. Available at:

<https://www.researchgate.net/publication/338411181_Marine_Application_of_Fiber_Reinforced_Composites_A_Review> [Accessed 4 June 2022].

Sabrowski, P. and Tschirschnitz, G., 2020. *Computational Fluid Dynamics Methods Explained.*

[online] Dive-solutions.de. Available at: <<https://www.dive-solutions.de/articles/cfd-methods>>

[Accessed 5 June 2022].

SimScale, 2019. *Eddy Viscosity | Boundary Conditions | SimScale.* [online] SimScale. Available at:

<<https://www.simscale.com/docs/simulation-setup/boundary-conditions/eddy-viscosity/>>

[Accessed 4 June 2022].

Sjodin, B., 2016. *What's The Difference Between FEM, FDM, and FVM?.* [online] Machinedesign.

Available at: <<https://www.machinedesign.com/3d-printing-cad/fea-and-simulation/article/21832072/whats-the-difference-between-fem-fdm-and-fvm>>

[Accessed 5 June 2022].

Sola, J., García-Baena, C. and Jesús, H., 2016. A REVIEW OF THE COMPUTATIONAL FLUID

DYNAMICS SIMULATION SOFTWARE: ADVANTAGES, DISADVANTAGES AND MAIN

APPLICATIONS. *Journal of Magnetohydrodynamics and Plasma Research*, 21(4), pp.417-424.

tanchain, 2022. *Typical application example of high-performance carbon fiber: marine ship field.*

[online] Tanchain. Available at:

<https://www.tanchaintex.com/news2/Typical_application_example_of_high_performance_carbon_fiber-en.html> [Accessed 4 June 2022].

Uddin, M., Hossen, M., Jahan, M. and Islam, M., 2021. Structural analysis of composite propeller

of ship using FEM. *Proceedings of the 13th International Conference on Mechanical Engineering*

(ICME2019), 1(1), p.7.

8.0 Works Cited

Wozniak, C.D., 2005. *Analysis, fabrication, and testing of a composite bladed propeller for a naval academy yard patrol (YP) craft*. NAVAL ACADEMY ANNAPOLIS MD. [online] pp.85-87. Available at: <<https://www.sciencedirect.com/science/article/pii/S002197971401056X>> [Accessed 4 June 2022].

Yamatogi, T., Murayama, H., Uzawa, K., Kageyama, K. and Watanabe, N., 2009. *Study on cavitation erosion of composite materials for marine propeller*. [online] ResearchGate. Available at: <https://www.researchgate.net/publication/287535959_Study_on_cavitation_erosion_of_composite_materials_for_marine_propeller> [Accessed 4 June 2022].

Yamatogi, T., Murayama, H., Uzawa, K., Mishima, T. and Ishihara, Y., 2011. Study on Composite Material Marine Propellers. *Journal of The Japan Institute of Marine Engineering*, [online] 46(3), pp.330-340. Available at: <https://www.jstage.jst.go.jp/article/jime/46/3/46_330/_pdf/-char/ja> [Accessed 4 June 2022].

Young, Y., Motley, M., Barber, R., Chae, E. and Garg, N., 2016. Adaptive Composite Marine Propulsors and Turbines: Progress and Challenges. *Applied Mechanics Reviews*, [online] 68(6), pp.1-29. Available at: <https://www.researchgate.net/profile/Eun-Jung-Chae/publication/308040839_Adaptive_Composite_Marine_Propulsors_and_Turbines_Progress_and_Challenges/links/5a0798460f7e9b68229b3b66/Adaptive-Composite-Marine-Propulsors-and-Turbines-Progress-and-Challenges.pdf> [Accessed 4 June 2022].

9.0 Appendix A – CFD Simulation for PIV Testing

For the CFD simulation, a computational domain split into two parts will be produced. One part will be the water tank region and will be created as a stationary part, and the second part will be the rotating propeller. The dimensions of the tank and propeller will match the physical tank and propeller (Nyongesa et al., 2022).

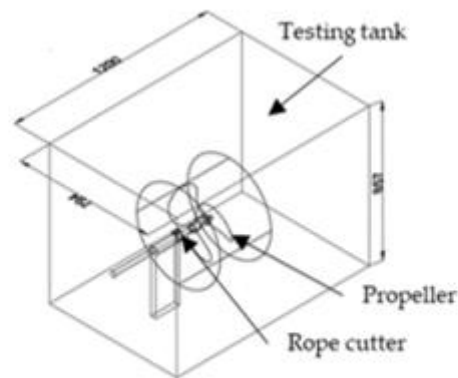


Figure 13: 3D model of the PIV Test Tank (Nyongesa et al., 2022, p.4)

A 3D mesh will be generated and the areas around the propeller will be refined to improve the accuracy of the fluid flow prediction.

The $k\epsilon$ model proposed by Launder and Spalding will be used for the flow simulation as it has proven to be robust and have good accuracy for a wide range of turbulent flow (Nyongesa et al., 2022).

The 3D dynamic system will be governed by the Navier-Stokes equations meaning that the Reynolds number will be used to predict the turbulence around the propeller (Nyongesa et al., 2022).

The next step would be to set the Boundary and Initial Conditions and process the simulations.

Post-processing mesh quality will then be analysed. Generally, three mesh resolutions are produced, then

the torque and thrust from the three solutions are graphically represented, and a maximum deviation will be attained at each propeller speed.

10.0 Appendix B

10.1 Displacement ratio consideration for small scale model

Since the model is scaled down, the displacement ratio would be smaller than the full-scale model. Therefore, to match the displacement rates and, in effect, the fluid flow around the propeller, the material used can be changed to one with a lower elastic modulus than the proposed prototype and alter the speed of the propeller to give the same displacement rate as the full-scale model (Hara et al., 2015).

$$\delta \propto \frac{L \cdot I^3}{E \cdot I} \propto \frac{\rho \cdot n^2 \cdot D^3}{E} \quad (1)$$

$$\frac{\delta}{D} \propto \frac{\rho \cdot n^2 \cdot D^2}{E} \quad (2)$$

where δ is the displacement, L the lift force, I the point of action, E the elastic modulus, I the second moment of area, ρ the density of fluid, n the number of rotations, D the diameter of propeller.

Figure 14: Displacement Ratio Formula (Hara et al., 2015)

The bending elastic modulus of model propeller E_m is derived from following equation (3)

$$E_m = \frac{\rho_m \cdot n_m^2 \cdot D_m^2}{\rho_a \cdot n_a^2 \cdot D_a^2} E_a \quad (3)$$

E_a is the bending elastic modulus of actual propeller, D_a the diameter of actual propeller, D_m the diameter of model propeller, n_a the diameter of propeller, n_m the number of rotations of model propeller, ρ_a the density of sea water ρ_m the density of water in cavitation tank.

Figure 15: Bending Elastic Moment Formula (Hara et al., 2015)

The above formula can give the bending elastic modulus needed for the scaled-down propeller model.

10.2 Reynolds Number consideration for small scale model

Reynolds number must also be considered.

(Density x velocity x diameter) / dynamic viscosity

The reduction in diameter needs to be compensated for by either increasing density or velocity or decreasing dynamic viscosity.

However, in PIV testing an increase in velocity will cause lower quality results from the validation experiment due to bubble formation; therefore, compromises will need to be made.