X International Conference on Computational Methods in Marine Engineering
MARINE 2023
J. Garcia-Espinosa, L. González, J.E. Gutiérrez & B. Serván-Camas (Editors)

Performance Prediction of KP505 Propeller Using Lifting Line Theory and RANS

Hammad Iftikhar*, Dr. Ali Can Takinaci†

* Faculty of Naval Architecture and Ocean Engineering
Istanbul Technical University,
İTÜ Ayazağa Campus, 34469 Maslak-ISTANBUL
E-mail: hammad0252@gmail.com

† Faculty of Naval Architecture and Ocean Engineering
Istanbul Technical University,
İTÜ Ayazağa Campus, 34469 Maslak-ISTANBUL
Email: takinaci@itu.edu.tr - Web page: https://web.itu.edu.tr/takinaci/

* Corresponding author: Hammad Iftikhar, hammad0252@gmail.com

ABSTRACT

Lifting line theory was developed over a century ago by Prandtl and has been employed to predict thrust and torque in various working conditions including lift of a wing, or as in this case, the thrust and torque produced by a propeller. Lifting line theory belongs to panel methods which are quick to converge as compared to CFD simulations. In lifting line theory, the actual blade geometry is replaced by span-wise panels of constant line circulation, which generates lift when it experiences an inflow. The method assumes the propeller blade sections to be replaced by a single line vortex that varies in strength from section to section.

CFD (Computational Fluid Dynamic) is one of the most popular and prolific technique used to solve hydrodynamic related problems nowadays. RANS (Reynolds-averaged Navier–Stokes) solver predicts the values of different performance indicators (Thrust, Torque, Efficiency) for the propeller and also give us valuable information about the flow field around the propeller.

This study aims to study the current trends in lifting line theory and CFD for propeller performance prediction and see how they compare to experimental results in accuracy and time to solution. Historically CFD has shown greater consistency with experimental results, but with new techniques being added to improve the accuracy of panel methods, the divide between the two methodologies might be small enough to bridge now.

Keywords: CFD; Numerical Simulation; Panel Methods; Ship Hydrodynamics; Propellers; Open Water

NOMENCLATURE

T	Thrust produced by propeller [N]
ρ	Fluid density [kg m^{-3}]
D	Diameter of propeller [m]
U_∞	Free stream velocity [m s^{-1}]
Q	Torque produced by propeller [Nm]
V_A	Advance Velocity [m s^{-1}]
n	Rotation speed [rad s^{-1}]
c	Chord Length [m]
J	Advance Ratio
C_L	Coefficient of Lift [-]
α	Angle of Attack
CFD	Computational Fluid Dynamics

1. INTRODUCTION

The global shipping industry plays a crucial role in the world economy, as it is responsible for the transportation of a vast array of goods, including raw materials, manufactured products, and perishable items. The industry has come under pressure in recent years due to rising fuel costs, overcapacity, and increasing environmental regulations. As a result, there has been a push to increase the efficiency of global shipping operations.

One way that the shipping industry is seeking to increase efficiency is through the use of technology. For example, many shipping companies are using advanced data analytics and visualization tools to optimize routes and reduce fuel consumption. Additionally, the use of digital twins, which are virtual replicas of ships, can help to identify problems and potential issues before they occur, leading to improved efficiency and reduced downtime.

There are also efforts being made to improve the efficiency of ports and terminal operations. This includes the use of automation and digitalization to streamline processes and reduce bottlenecks. Additionally, there is a push to adopt more sustainable practices in ports and terminals, such as using renewable energy sources and reducing waste.

Finally, there are ongoing efforts to improve supply chain efficiency through the use of collaborative initiatives and partnerships. For example, many companies are working together to share resources and create more efficient and cost-effective logistics networks.

Another way that the shipping industry is increasing efficiency is through the use of more fuel-efficient vessels. These ships are designed to be lighter and more aerodynamic, which can help to reduce fuel consumption and emissions. Additionally, many shipping companies are exploring the use of alternative fuels, such as liquified natural gas (LNG), to power their vessels. Furthermore, improving the propeller and prime mover performance can lead to significant improvement in ship's efficiency and can lead to major gains in fight against climate change.
Propeller design is critical to the efficiency and performance of ships. A properly designed propeller can significantly reduce fuel consumption, reduce noise and vibration, and increase the speed and maneuverability of the vessel.

One key factor in propeller design is the number of blades. A propeller with fewer blades may be more efficient at high speeds, but it can also be less efficient at low speeds and may create more noise and vibration. On the other hand, a propeller with more blades may be more efficient at low speeds, but it may be less efficient at high speeds and may be more vulnerable to damage. Another important factor in propeller design is the blade shape. Blades that are too thin may be prone to bending and breaking, while blades that are too thick may be less efficient. The angle of the blades, or pitch, is also important. A propeller with a higher pitch may be more efficient at high speeds, but it may be less efficient at low speeds and may be more prone to cavitation, which is the formation of bubbles on the surface of the blade.

Propeller design is also influenced by the size and power of the ship. Larger vessels with more powerful engines will require larger and more robust propellers, while smaller vessels with less powerful engines will require smaller and more efficient propellers.

In conclusion, propeller design is a critical factor in the efficiency and performance of ships. The number of blades, blade shape, and pitch all play a role in determining the efficiency and effectiveness of a propeller. Properly designed propellers can significantly reduce fuel consumption and improve the speed and maneuverability of a vessel.

The research focuses on using different methods i.e., lifting line theory and RANS CFD simulation to predict the performance of KP505 propeller. The KP505 was designed by Korea Research Institute of Ships and Ocean Engineering (KRISO) to be used for the KRISO Container Ship (KCS).

Propeller open water characteristics are useful indicators of the performance for propellers in undisturbed uniform flows with steady loads. The thrust and torque, T and Q, can be normalized as:

$$\mathrm{K_T} = \frac{\mathrm{T}}{\rho \mathrm{n}^2 \mathrm{D}^4} \tag{1}$$

$$\mathrm{K_Q} = \frac{\mathrm{Q}}{\rho \mathrm{n}^2 \mathrm{D}^5} \tag{2}$$

The thrust and torque coefficients, K_T and K_Q, are generally plotted against a range of advance coefficient, J, which is defined as:

$$\mathrm{J} = \frac{\mathrm{V_A}}{\mathrm{nD}} \tag{3}$$

2. LITERATURE REVIEW

Jarle V. Kramer, John Martin K. Godø, Sverre Steen compared results from lifting line method and CFD for a hydrofoil presented in Numerical Towing Tank Symposium in 2018. They concluded that lifting line method is both simple to implement and faster to execute on a computer as compared to CFD. They tested variations of wings made from NASA LS417 foil profile, and overall found reasonable correlation between CFD and lifting line theory however, they did note that lifting line theory diverged from experimental results when sweep was introduced into the wing geometries. (Kramer, 2018)

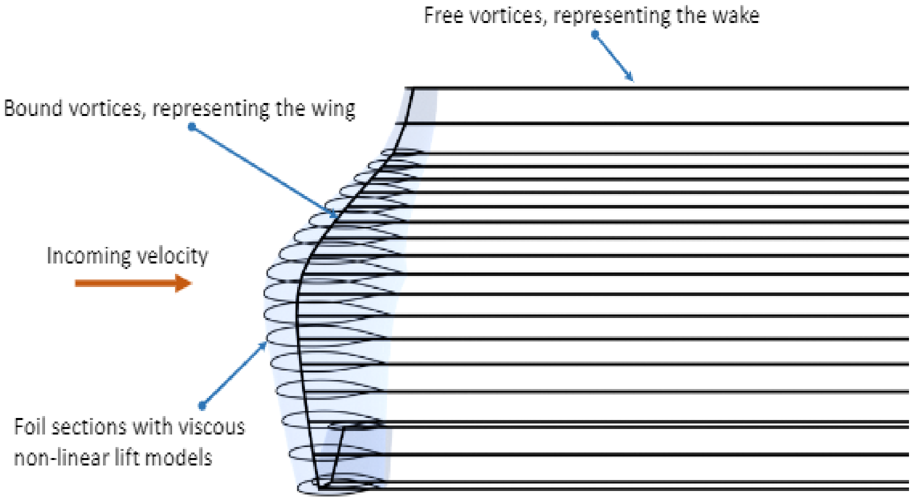

Figure 1. Hydrofoil Sections

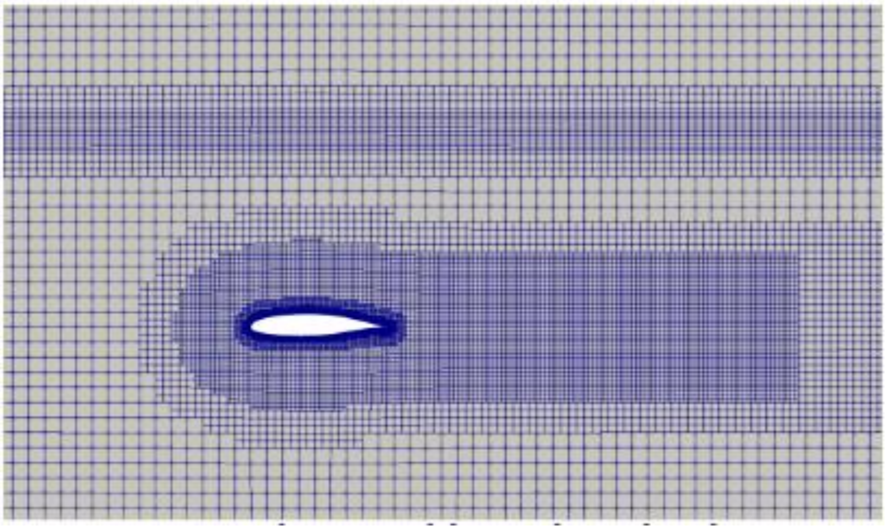

Figure 2. Hydrofoil Mesh

Brenden P. Epps and Richard W. Kimball presented a unified lifting line method for the design and analysis of axial flow propellers and turbines. The method incorporates significant improvements to the classical lifting line methods for propeller design to extend the method to the design of turbines. (B. Epps, 2013) (Epps, 2016)

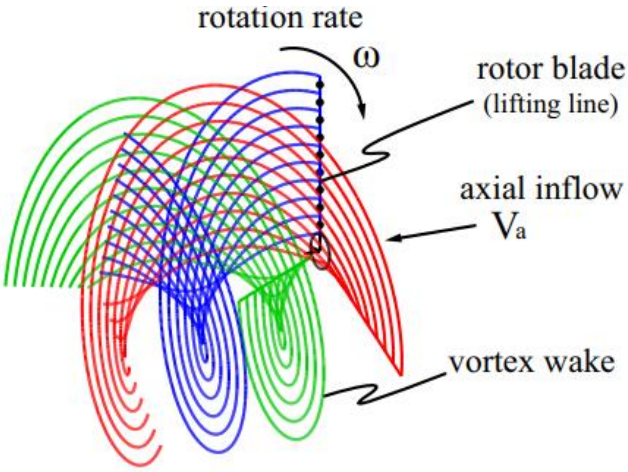

Figure 3. Lifting Line Representation

Jadmiko et. al, used OpenProp and CFD to design the propeller for their vessel named Jalapith 3 for Solar Sport One. They found good correlation between CFD and Openprop results at higher advance ratios but the results varied on lower advance ratios. CFD predicted higher thrust and torque values as compared to lifting line theory. (Edi Jadmiko, 2020)

The basis of almost all of finite wing theory is based on Prandtl's lifting line theory. The theory approximates the flow around a wing by using horseshow shaped vortices, he accomplished this by using the Kutta-Juokowski theorem along with spanwise lift (Anderson, 2011):

$$\rho V_\infty \Gamma = \frac{1}{2}\rho V_\infty^2 C_L(\alpha)c \tag{4}$$

3. PROPELLER CHARACTERISTICS

The KP505 was designed by Korea Research Institute of Ships and Ocean Engineering (KRISO) to be used for the KRISO Container Ship (KCS). The propeller has been extensively tested and sufficient data is available related to its performance and characteristics to easily compared calculated results with accurate data points.

Table 1. Propeller Characteristics

Propeller Type	FPP	**Ae/Ao**	0.800
No. of Blades	5	**Rotation**	Right Hand
Diameter (m)	0.25	**Hub Ratio**	0.18
P/D (mean)	0.950	**Section**	NACA66

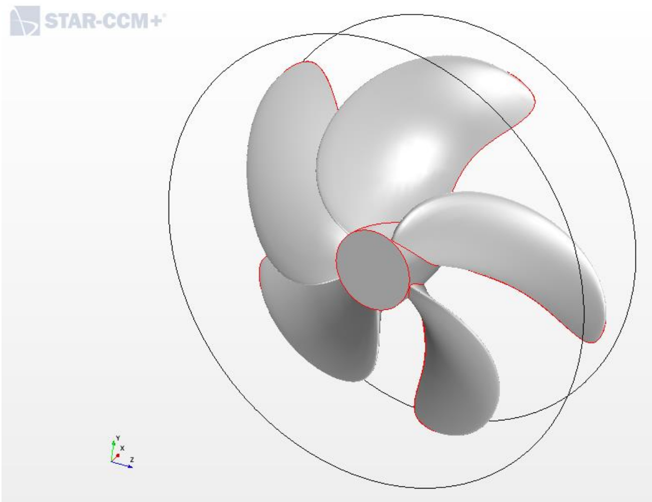

Figure 4. Propeller Geometry

4. COMPUTATIONAL FLUID DYNAMICS ANALYSIS

In computational fluid dynamics (CFD), a moving reference frame is a frame of reference that is in relative motion to the fluid being analyzed. This can be useful when studying the flow of fluids around moving objects, such as ships, airplanes, or vehicles.

Using a moving reference frame allows the fluid flow to be analyzed relative to the moving object, rather than in an absolute reference frame. This can make it easier to analyze the flow patterns and forces acting on the object, as the effects of the object's motion are accounted for in the reference frame.

To use a moving reference frame in CFD, the equations of motion for the fluid flow are transformed into the moving reference frame. This involves adding a term to account for the relative velocity between the fluid and the reference frame. The transformed equations are then solved using CFD methods, such as finite element analysis or finite volume analysis.

One challenge in using a moving reference frame in CFD is that the grid used to discretize the flow field must also be transformed into the moving reference frame. This can be complex and time-consuming, especially for complex geometries and flow conditions. However, the use of a moving reference frame can greatly improve the accuracy and realism of CFD simulations for problems involving moving objects.

However, the complexity of this method is balanced by the ease it provides in allowing for complex motion characteristics to be studied in steady state analysis instead of unsteady analysis. The application of moving reference frame instead of moving mesh allows for unsteady motion to be studied in a steady state analysis without the inclusion of timesteps and allows for a much faster and much simpler analysis with results of comparable accuracy.

Velocity inlet is used whose magnitude is calculated according to advance ratio (J). As the diameter and rotation speed of propeller has been fixed, advance ratio is dependent solely on the velocity.

Pressure outlet with pressure at infinity is used. The walls are selected as no-slip condition and sufficient distance is kept from the walls to the propeller to ensure to blockage effect.

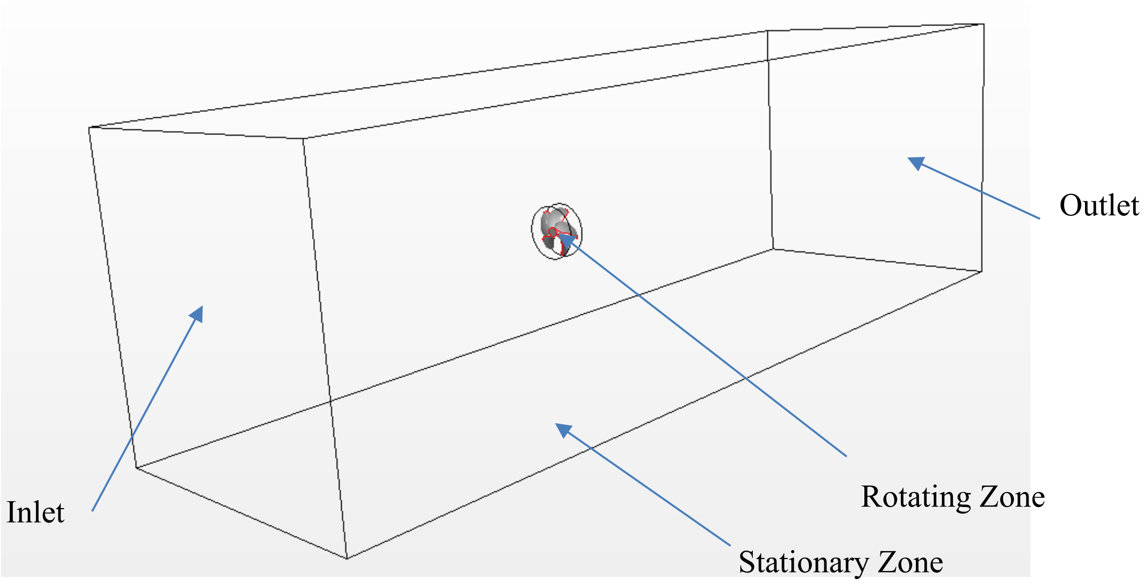

Figure 5. Boundary Conditions for CFD

4.1. Mesh

In computational fluid dynamics (CFD), the mesh is a discretization of the flow field into small, interconnected elements. The proper construction and quality of the mesh is important for several reasons.

One reason is accuracy. A properly constructed mesh will capture the important features and details of the flow field, and will provide accurate results. A poorly constructed mesh, on the other hand, may miss important features or introduce errors, leading to inaccurate or misleading results.

Another reason is efficiency. A well-constructed mesh will use the minimum number of elements necessary to capture the important features of the flow field, while also avoiding unnecessary refinement in regions where it is not needed. This can significantly reduce the computational cost of the CFD simulation and allow it to be completed in a reasonable amount of time.

A third reason is the ability to accurately capture the boundary conditions of the problem. The mesh must be constructed in such a way that the boundary conditions are correctly enforced, otherwise the results of the simulation may be invalid.

In conclusion, the proper construction and quality of the mesh is critical for the accuracy, efficiency, and validity of CFD simulations. Proper mesh construction is an important aspect of the CFD process, and requires careful consideration and attention to detail.

Cut-cell mesh is generated with small, fine elements near the propeller and the rotating zone and large coarse particles in the rest of the stationary domain. The rotating domain contains around 2.5 million elements and the stationary domain around 1.1 million.

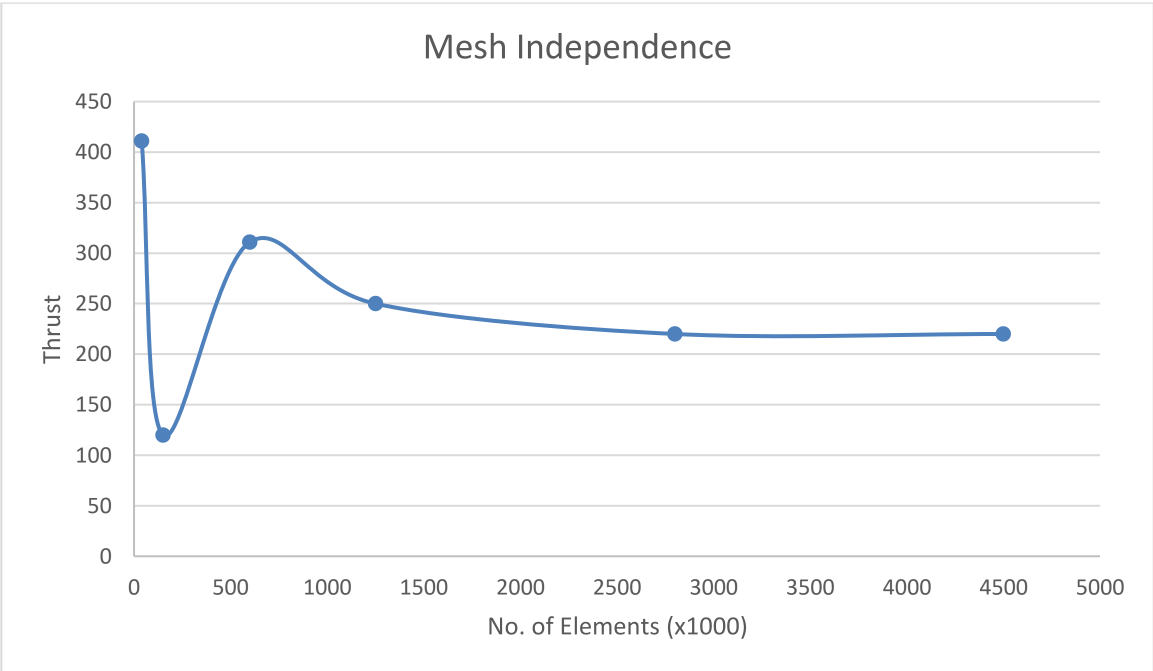

Figure 6. Mesh Independence

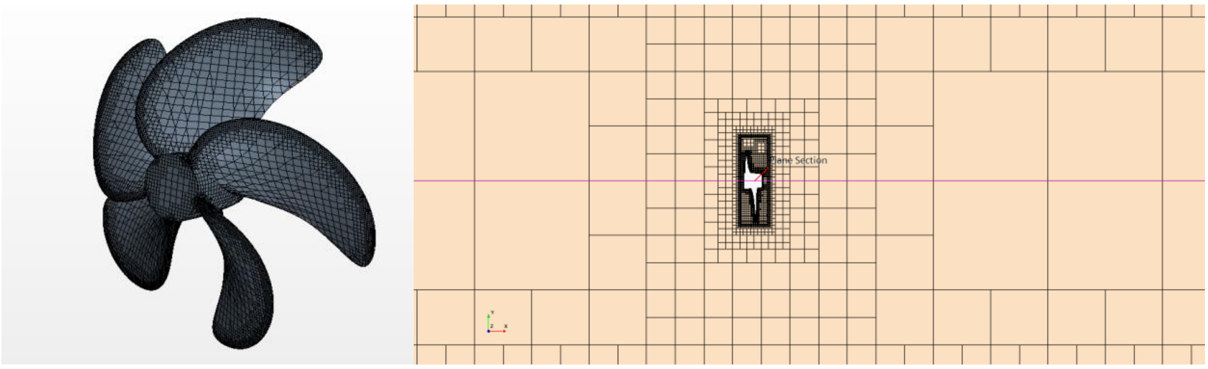

Figure 7. Propeller and Domain Mesh

5. RESULTS AND DISCUSSION

The results from CFD are very promising and match well with experimental data over a wide range of Advance Ratios. Thrust prediction for low Advance Ratio is very accurate with most of the performance parameters being within 5% of experimental values. Similarly, for Torque coefficients the results match very well with experimental data and other than a couple of points towards the higher end of Advance Ratios the results are within single digit error percentage.

Table 2. Comparison of CFD and Experimental Results

	CFD		Experimental		Error	
J	**KT**	**10KQ**	**KT**	**10KQ**	**KT**	**10KQ**
0.1	0.479	0.644	0.494	0.683	3.06%	5.73%
0.2	0.432	0.590	0.447	0.636	3.24%	7.22%
0.3	0.382	0.554	0.392	0.585	2.64%	5.32%
0.4	0.330	0.500	0.343	0.530	3.73%	5.72%
0.5	0.278	0.473	0.290	0.471	4.23%	0.50%
0.6	0.225	0.414	0.241	0.409	6.49%	1.26%
0.7	0.173	0.356	0.193	0.342	10.31%	4.01%
0.8	0.120	0.288	0.141	0.270	15.17%	6.66%
0.9	0.063	0.152	0.087	0.192	27.60%	20.89%
1	0.030	0.094	0.029	0.108	2.27%	13.25%

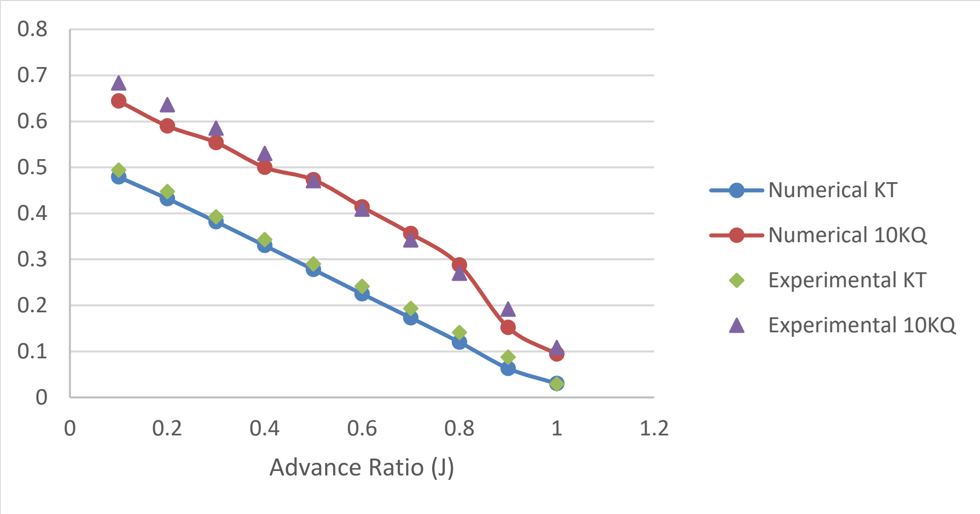

Figure 8. Comparison of CFD and Experimental Results

For Lifting Line Theory, we see that the thrust results are very close to experimental values and considering the simplicity and time taken for this method, it is significant how close to experimental values these results are. However, on the Torque side we see a marked deviation from experimental results. This can be attributed to lack of viscous model in Lifting Line Theory, which leads to a big difference in Torque prediction. (Angga Septiyana, 2020)

Table 3. Comparison of LLT and Experimental Results

	Lifting Line Theory		**Experimental**		**Error**	
J	**KT**	**10KQ**	**KT**	**10KQ**	**KT**	**10KQ**
0.3	0.374	0.478	0.392	0.585	4.59%	18.29%
0.4	0.335	0.444	0.343	0.530	2.33%	16.23%
0.5	0.292	0.402	0.290	0.471	-0.69%	14.65%
0.6	0.246	0.352	0.241	0.409	-2.07%	13.94%
0.7	0.198	0.293	0.193	0.342	-2.59%	14.33%
0.8	0.146	0.225	0.141	0.270	-3.55%	16.67%
0.9	0.094	0.151	0.087	0.192	-8.05%	21.35%
1	0.041	0.070	0.029	0.108	-41.38%	35.19%

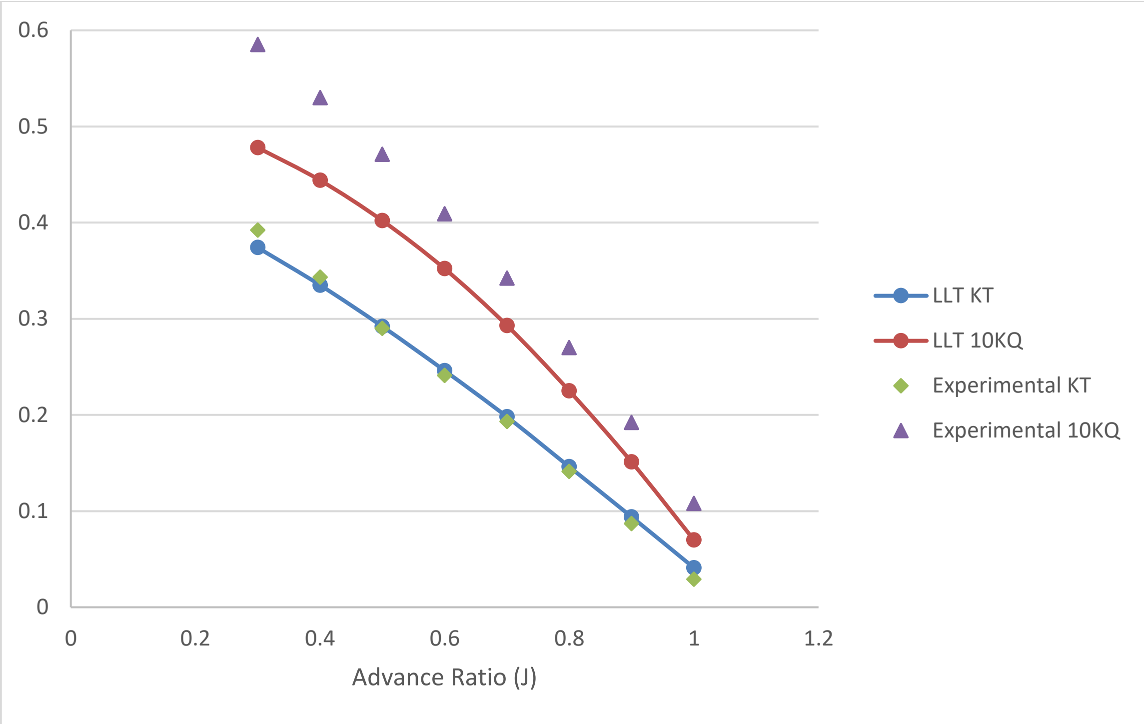

Figure 9. Comparison of LLT and Experimental Results

Side-by-side comparison of the two techniques leads to some interesting results. In this particular case, for thrust prediction Lifting Line Theory offers better results and closer approximation to the experimental values as compared to CFD. Furthermore, with both the techniques we see a marked increase in error values with increasing Advance Ratio and this can be explained by the increasing non-linearity and turbulence associated with increased Advance Ratio.

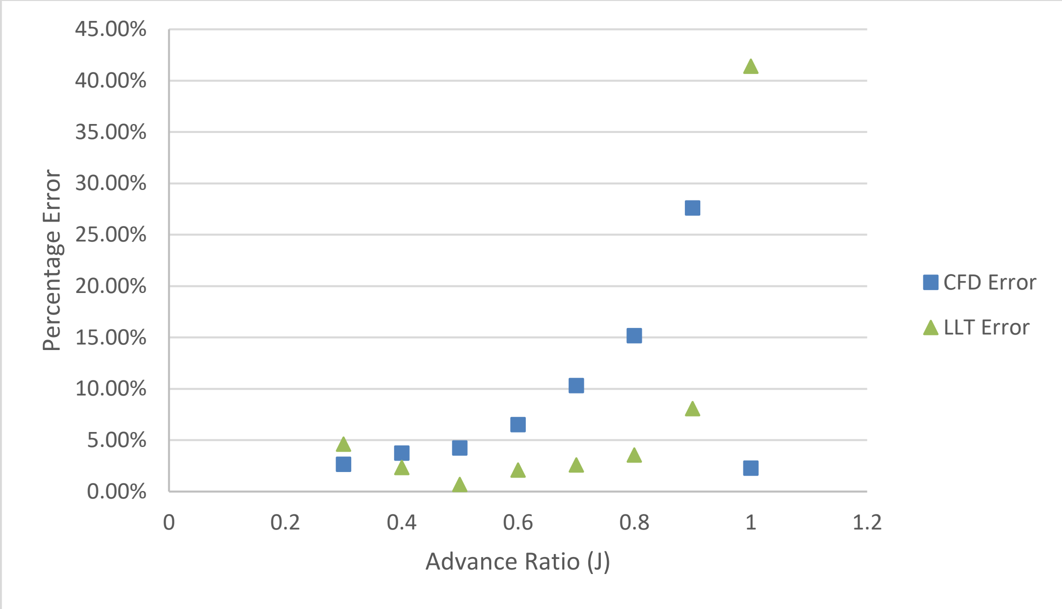

Figure 10. Error Comparison in Thrust

Through side-by-side comparison of Torque values, the advantage of CFD is clearly visible and the problem with fast acting and simple methodology of Lifting Line Theory is glaringly obvious. CFD offers much better prediction for torque coefficient as compared to Lifting Line Theory and where in some cases the difference is more than 20% between CFD and Lifting Line Theory results.

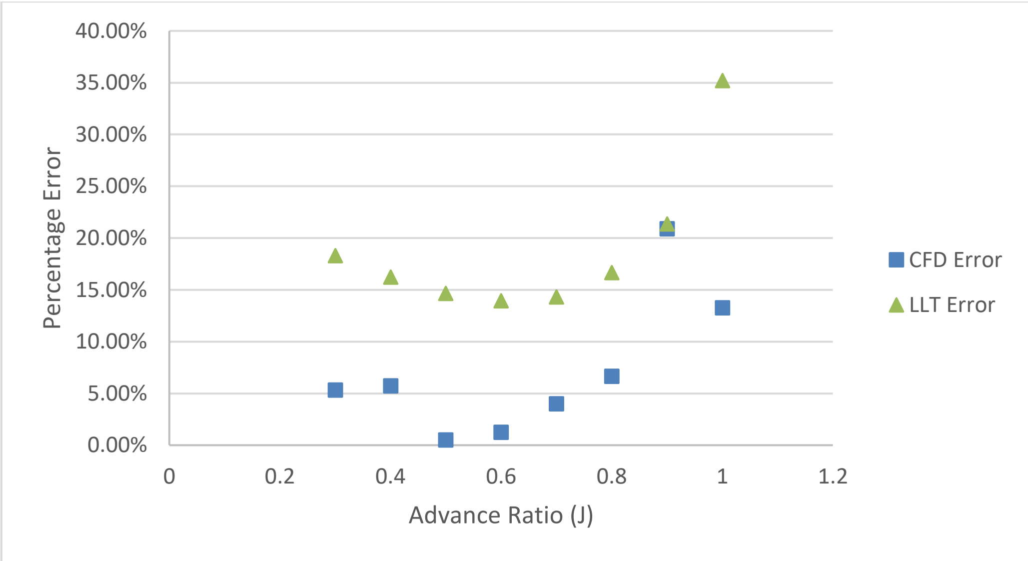

Figure 11. Error Comparison in Torque

6. CONCLUSION

Lifting Line Theory and Computational Fluid Dynamics (CFD) are both techniques used to analyze and predict the flow of fluids, such as air or water. However, there are several key differences between the two approaches.

One major difference is the level of detail and accuracy of the models. Lifting Line Theory is generally less accurate and less detailed than CFD, as they use a simplified representation of the flow field. Lifting Line Theory is based on the idea of dividing the surface of the object being analyzed into a series of flat panels, and solving for the flow properties around each panel. This can provide useful results for certain types of problems, but it is not as accurate as a full CFD simulation, which solves the Navier-Stokes equations for the entire flow field.

Another difference is the type of problems that each method is well suited for. Lifting Line Theory is generally best for problems involving bodies of revolution, such as cylinders and spheres, and for simple geometries with low levels of complexity. CFD, on the other hand, can handle more complex geometries and flow conditions, and is generally more accurate for predicting the flow around arbitrary shapes.

A third difference is the computational cost and required resources. Lifting Line Theory is generally much faster and require less computational power than CFD simulations. This makes it well suited for problems that need to be solved quickly, or for use in real-time applications. However, the trade-off is that Lifting Line Theory is less accurate and less versatile than CFD.

Overall, Lifting Line Theory and CFD are two different approaches to analyzing and predicting fluid flow, each with its own strengths and limitations. Lifting Line Theory is generally faster and simpler, but less accurate and less versatile, while CFD is more accurate and versatile, but also more computationally intensive.

REFERENCES

Anderson, J. D. (2011). Incompressible Flow over Finite Wings. In *Fundamentals of Aerodynamics* (5th ed., pp. 411-444). New York: McGraw Hill.

Angga Septiyana, K. H. (2020). Comparative Study Of Wing Lift Distribution Analysis Using Numerical Method. *Jurnal Teknologi Dirgantara, 18*(2), 129-139.

B. Epps, A. R. (2013). Unified Rotor Lifting Line Theory. *Journal of Ship Research, 57*(4), 1-21.

Edi Jadmiko, e. a. (2020). Analysis Symmetrical Blade Propeller Performance for Jalapatih 3 Ship Using CFD. *International Journal of Marine Engineering Innovation and Research, 5*(4), 216-223.

Epps, B. (2016). On the rotor lifting line wake model. *Journal of Ship Production and Design, 32*(3), 1-15.

Kramer, S. S. (2018). Hydrofoil Simulations - Non-linear Lifting Line vs CFD. *Numerical Towing Tank Symposium.* Cortona.