

NUMERICAL STUDY OF THE HYDRODYNAMIC PERFORMANCE OF RIM-DRIVEN PROPULSION FEATURING A MODIFIED 19A DUCT

Meysam Masoumpour

Department of Maritime Engineering, Amirkabir University of Technology, Tehran, Islamic Republic of Iran

Hassan Ghassemi* 

¹⁾ Department of Maritime Engineering, Amirkabir University of Technology, Tehran; Islamic Republic of Iran

²⁾ Int. School of Ocean Science and Engineering, Harbin Institute of Technology, Weihai, China

S. Hossein Mousavizadegan

Department of Maritime Engineering, Amirkabir University of Technology, Tehran, Islamic Republic of Iran

Guanghai He

Harbin Institute of Technology, China

* Corresponding author: gaseemi@aut.ac.ir (Hassan Ghassemi)

ABSTRACT

The rim-driven propeller (RDP) is an innovative propulsion system that is primarily used in underwater vehicles and the bow thrusters of ships. In this study, the Reynolds-averaged Navier-Stokes (RANS) equations are employed together with the moving reference frame method and steady-state numerical simulations to address challenges related to applicability. The SST turbulence model is also incorporated. Initially, a Ka-Series+19A ducted propeller (DP) is considered, and the numerical results for its hydrodynamic performance are found to show a close correlation with experimental data. Notably, the thrust coefficient of the duct at low advance coefficients is high, indicating that the duct can operate efficiently under heavy load conditions. The study then focuses on the RDP, which uses the same propeller but features a distinct duct design due to its rim-driven configuration. The hydrodynamic open-water characteristics of the RDP are obtained and compared with those of the DP. The results reveal that the RDP has lower efficiency than the DP, primarily due to the gap and the presence of the rotor in the RDP. Furthermore, a detailed analysis of the pressure distribution on the surfaces of the blade and duct is presented, as well as the velocity and pressure contours at various downstream positions for both the DP and RDP. Particular attention is paid to the flow gap between the propeller and duct, along with the associated turbulence intensity.

Keywords: Rim-driven propulsion, Hydrodynamic performance, Kaplan propeller, Pressure coefficient, Modified duct 19A.

INTRODUCTION

The limitations of traditional ship propulsion technologies are becoming increasingly apparent. As the demand for propulsion power grows, larger and more powerful main engines are required; moreover, the structural design of the propulsion shaft system, which typically includes the engine, bearings, gearbox, seals, strut and shaft, has become more complex. This increased complexity not only raises the costs associated with ship design and construction but also reduces the available space on the vessel, ultimately decreasing the propulsion efficiency [1]. The concept of a rim-driven propeller (RDP) emerged as a response to the limitations of traditional propulsion systems. Over the past two decades, the

maritime industry has increasingly focused on electrification and the development of hybrid vessels, positioning the RDP as a revolutionary innovation in ship electric propulsion. The RDP represents a novel propulsion system that eliminates the need for the construction of a conventional shaft: instead, it employs a compact and integrated structure characterised by a high motor power density. The system connects the driving engine to a shroud that encircles the propeller, making the propulsion rim-driven rather than shaft-driven, as seen in traditional systems. This innovative approach not only optimises the use of space but also enhances propulsion efficiency. In particular, this design optimises the use of hull space, resulting in a more spacious engine room and enabling a more flexible arrangement of onboard equipment, meaning

that the ship becomes significantly more versatile. Moreover, engineers can improve the energy efficiency by adjusting the number of auxiliary engines connected to the power grid according to the actual navigation needs. In addition, the RDP provides 360° manoeuvrability, which reduces the ship's turning diameter and greatly improves its handling capabilities [2].

One of the most significant advantages of this system is the absence of a shafting mechanism, which, combined with the elimination of tip clearance, prevents the formation of tip vortices and reduces noise, thereby enhancing thrust levels. The compact and lightweight design of the system also makes it particularly suitable for applications such as underwater vehicles, including remotely operated vehicles (ROVs), autonomous underwater vehicles (AUVs), and bow and aft thrusters [3], ultimately improving their overall performance.

An RDP is an innovative type of combined motor propulsor, commonly referred to as a shaftless propeller. Unlike conventional propellers driven by a shaft, the RDP uses a circular rim connected to the blade tips to generate torque. A motor stator is positioned within the duct or nozzle, while a permanent magnet (PM) rotor is housed within the duct surrounding the rotor, as illustrated in Fig. 1(a). To facilitate cooling, the stator and rotor are separated by a gap and submerged in seawater. Furthermore, the RDP does not require waterproofing during rotation, as can be seen from the internal configuration in Fig. 1(b)

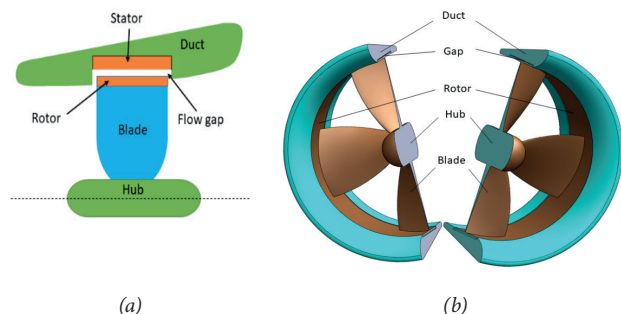


Fig. 1. (a) Integrated design mechanism; (b) internal arrangement of an RDP.

Compared to conventional shaft-driven propellers, RDPs show different behaviour in open water, stemming from the direct connection of the permanent magnet motor rotor to the blade, which replaces the conventional propeller tip. The unique performance of an RDP with a hub, combined with its specialised rotor design, significantly influences the interaction between the rim and duct, thus affecting the local fluid flow field as well as the overall thrust and torque. As a result, an RDP offers a more efficient and quieter propulsion solution, primarily due to the elimination of components such as gearboxes and shafts. The potential of this device to reduce environmental impact also positions it as a promising choice for future applications.

An RDP is most comparable to a DP in terms of its construction and operation, as the propeller blades are

mounted on a rotating rotor within a duct-shaped structure. However, unlike the DP, which relies on a shaft system for propulsion, the RDP operates without one. Again, in contrast to a DP, an RDP does not have a tip clearance, with the duct and rotor surfaces instead creating a gap for fluid flow. Both the RDP and DP have similar hydrodynamic characteristics, resulting in high-efficiency and low-noise propulsion [4, 5]. Some researchers have used experimental findings from a DP to confirm the accuracy of numerical simulations, due to the structural similarities between a DP and RDP [6].

The first conceptual model of the RDP was patented by Kort [7]. A notable advantage of RDPs is the elimination of the wake effects generated by the fairwater and rudder, resulting in uniform inflow and lower noise levels [8]. RDPs can be installed in vertical or horizontal through-body tunnels at both the front and rear of the vehicle, enhancing manoeuvrability [9]. It is also vital to synchronise the hydrodynamic performance of an RDP with relevant parameters to achieve optimal efficiency. Factors influencing the hydrodynamic efficiency include the structural design of the propeller, the gap thickness [10], and the size and shape of the duct, among others [11].

Numerous studies have explored various aspects of RDP design and performance [12-14]. These investigations have often been undertaken in the form of physical experiments, numerical simulations, and combined methods [15, 16]. A ring thruster propulsion system was experimentally studied to determine the relevant hydrodynamic performance [17]. Although RDPs offer significant hydrodynamic and structural advantages compared to traditional propellers, as a relatively new propulsion technology, they still present challenges in regard to their design and performance analysis that require further research.

In recent years, Reynolds-averaged Navier–Stokes (RANS) methods have been used to predict the hydrodynamic performance of RDPs; however, research on this topic has focused on a limited number of possible geometries and operating conditions. Numerical simulations provide a good approximation for iterative design optimisation at a low computational cost, and the vortex effect at the blade tip has been studied numerically [18, 19]. Some key factors such as the aspect ratio of the duct, the diffusion ratio, contraction ratio of the duct, and blade tip diameter ratio have been investigated, as these impact on the hydrodynamic performance of the RDP [20]. The RANS equations with the shear stress transport (SST) $k-\omega$ turbulence model have been applied to simulate the RDP-stator interactions at relatively low advance coefficients compared to the re-normalisation group (RNG $k-\epsilon$) model [21, 22]. Numerical simulations of RDP propellers have made extensive use of these models [23-25], and have also utilised the moving reference frame (MRF) approach, a steady-state approximation for analysing situations in which domains rotate relative to one another, where the flow in each domain is solved with the MRF equations. The hydrodynamic performance of three different RDPs with uniform diameters and various blade numbers was analysed using a RANS solver and an initial blade design

approach [26]. The researchers developed a simulation-based design optimisation approach to enhance the RDP blade shape, to improve the propulsion performance and reduce cavitation. The external characteristics were studied through experiments, and numerical simulations were carried out to study the inner flow characteristics [27]. A block diagram of the computer system was created for the design of ducted propellers, which included all of the components necessary for the correct design process for propellers operating in ducts with a selected geometry [28, 29]. The interaction between the rotor and duct, and the flow-structure interactions, have recently been studied using advanced FSI algorithms [30].

Numerous research articles have focused on RDPs, with particular reference to electric motors, control technologies, and hydrodynamic performance. These studies provide valuable references for the design and numerical analysis of RDPs. In this paper, we emphasise the need to analyse velocity and pressure, as these factors significantly impact hydrodynamic performance. A comprehensive flow-field analysis is conducted here, which highlights the effects of gap and turbulence. It is important to revise the recommendations for propeller design, since the blade loading in an RDP differs from that of a traditional propeller, and the rim alters the flow interactions. Consequently, challenges persist in regard to the hydrodynamic design and performance analysis of RDPs. This study advances research on the topic of RDPs by systematically comparing the hydrodynamic performance of a conventional propeller DP and an RDP, using identical Ka4-70 propeller blades and a modified 19A duct. Unlike previous studies that have focused on isolated RDP analyses, this direct comparison highlights the effects of rim-driven configurations on the flow dynamics, pressure distribution, and turbulence. In addition, the modified duct design is analysed in terms of its impact on gap flow and efficiency losses, a critical yet underexplored area of RDP optimisation.

This article describes numerical simulations carried out to model the hydrodynamic performance of the RDP, focusing on the adjustment, modification, and distribution modelling of the Ka-series blades and 19A duct. The numerical results for the DP were compared with experimental data, and showed good agreement. The RDP is a novel development in which the rotor is driven by a permanent magnetic motor. The rotor is therefore connected to the inside of the duct to give a compact device without mechanical transmission loss, no tip vortex, and reduced noise and vibration. In this study, the hydrodynamic performance of the DP and RDP was explored, and the effect of rim-driven propulsion on the duct and blade tip was investigated. The rest of this paper is structured as follows: Section 2 introduces the governing equations and expressions for the hydrodynamic performance. Section 3 describes the numerical simulations and validation of the DP. The hydrodynamic performance of the RDP, the pressure distribution, the velocities at different downstream locations, and the flow gap between the propeller and duct, along with the associated turbulence intensity (TI), are discussed in Section 4. Finally, a conclusion is given in Section 5.

GOVERNING EQUATIONS

In this section, a computational fluid dynamics (CFD) solution is derived from the fundamental governing equations of hydrodynamics, which consist of the continuity equations and the momentum conservation equations. Ansys CFX software was used as the solver to compute the equations given below.

$$\text{Continuity equation} \quad \frac{\partial \bar{u}_i}{\partial x_i} = 0 \quad (1)$$

$$\text{Momentum equation} \quad \rho \left(\frac{\partial \bar{u}_i}{\partial t} + \bar{u}_j \frac{\partial \bar{u}_i}{\partial x_j} \right) = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j} \left[\mu \left(\frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i} \right) - \rho \overline{(u'_i u'_j)} \right] \quad (2)$$

where t is the flow time; u_i, u_j ($i, j = 1, 2, 3$) are the average velocity components; ρ is the density of water; p is the pressure; μ is the dynamic viscosity of water; and $-\rho \overline{(u'_i u'_j)}$ represents the Reynolds stress. These equations are solved using CFD software. The pressure dissipative term is solved using a second-order central difference scheme, the convective term is solved using a second-order upwind scheme, and velocity-pressure decoupling is carried out using a coupled algorithm with a standard turbulence model.

TURBULENCE MODEL

The SST $k-\omega$ turbulence model is added to complete the set of governing equations. This model combines the original $k-\omega$ formulation, which provides a robust near-wall treatment, with the capability to accurately predict flows under moderate adverse pressure gradients. The model includes two additional transport equations to represent the turbulent kinetic energy (k) and the specific dissipation rate (ω). The transport equations for k and ω are detailed in [21], and an overview will be presented here. The SST $k-\omega$ model was selected for its accuracy in capturing adverse pressure gradients and flow separation near walls, which is important for modelling the gap flows in RDP.

HYDRODYNAMIC PERFORMANCE

The open water hydrodynamic performance is evaluated here using the realisable SST $k-\omega$ turbulence model, in which the performance is calculated by solving these equations within the solution domain, and pressure and shear stress are incorporated to determine the hydrodynamic performance coefficients for the propeller. These equations are solved in conjunction with Eqs. (1) and (2). The MRF approach is validated against experimental data on the DP, and the computational cost is balanced with the steady-state accuracy for rotating systems. The hydrodynamic coefficients considered here include the advance coefficient, thrust coefficient, torque coefficient, efficiency, pressure coefficient and resultant velocity, and are defined as follows:

$$\text{Advance coefficient} \quad J = \frac{V_a}{n D} \quad (3)$$

Thrust
coefficient
of propeller

$$K_{tp} = \frac{T_p}{\rho n^2 D^4} \quad (4)$$

Thrust
coefficient
of duct

$$K_{td} = \frac{T_d}{\rho n^2 D^4} \quad (5)$$

Torque
coefficient
of propeller

$$K_q = \frac{Q}{\rho n^2 D^5} \quad (6)$$

Efficiency
propeller

$$\eta = \frac{J}{2\pi} \frac{K_{tT}}{K_q} \quad \text{where } K_{tT} = K_{tp} + K_{td} \quad (7)$$

Pressure
coefficient

$$C_p = \frac{p - p_\infty}{0.5 \rho V_R^2} \quad (8)$$

Resultant
velocity

$$V_R = \sqrt{V_a^2 + (2\pi r n)^2} \quad (9)$$

where V_a is the advance velocity in m/s ; n is the rotational velocity of the propeller in revolution per second (RPS); ρ is the fluid density in kg/m^3 ; D is the diameter of the propeller in m ; and J is the advance coefficient. K_{tp} , T_p , K_{td} , T_d , K_q , Q , and η are the thrust coefficient of the propeller, propeller thrust, thrust coefficient of the duct, duct thrust, torque coefficient, torque, and efficiency, respectively. C_p represents the pressure coefficient, while p and p_∞ denote the local pressure and reference pressure (both in Pascals), respectively. V_R is the representative resultant velocity of the propeller, and r is radius over which the velocity is obtained.

MODEL SETTINGS AND VALIDATION OF NUMERICAL SIMULATION STRATEGY

This section discusses the division of the calculation domain and the grid used for both the DP and RDP simulations under open water conditions. Due to a lack of specialised experimental data for the RDP, it is essential to validate the RDP simulation carefully. To ensure the accuracy and reliability of the CFD calculation method, a hydrodynamic performance analysis of the 19A + Ka4-70 DP model was first conducted, using experimental data as a benchmark.

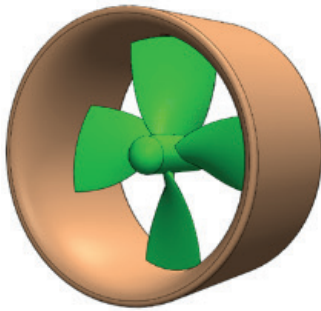


Fig. 2. Dimensional model of the ducted propeller

Tab. 1. Main parameters of the Ka4-70 propeller and duct

Parameter	Value	Parameter	Value
Blade diameter (D)	0.25 [m]	Duct type	19A
Number of blades	4	Duct length	0.67D
Pitch ratio (P/D)	1.0	Duct diameter	1.237D
Expanded area ratio (EAR)	0.7	Rotational velocity (RPS)	10

The simulation results for the DP are compared with experimental data (from Wang et al. [31]) to validate our CFD calculation approach for the RDP, as the flow fields of the two propellers are similar. A general model of the Ka4-70 DP is shown in Fig. 2, and the detailed specifications of the Ka4-70 propeller are presented in Table 1. The gap between the tip of the propeller blade and the inner wall of the duct is 1.0 mm.

PHYSICAL MODEL

The structural layout of the RDP is similar to that of the DP, as the Ka4-70 DP design is employed in this investigation. Table 2 presents some of its key characteristics, while Fig. 3 shows a 3D image of the RDP. The primary distinction between the DP and RDP lies in the presence of a rotor surrounding the blade in the RDP.

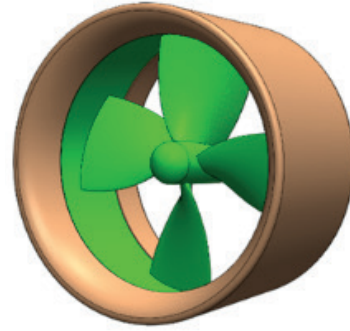


Fig. 3. Dimensional model of the RDP

Tab. 2. Geometrical details of the RDP

Parameter	Value	Parameter	Value
Number of blades (Z)	4	Pitch ratio (P/D)	1.0
Blade diameter (D)	0.25 [m]	Hub ratio	0.167
Expanded area ratio (EAR)	0.7	Rotational velocity (RPS)	10
Rotor diameter (m)	0.251	Rotor thickness (m)	0.001

NUMERICAL MODEL

The RANS equations are employed here to simulate the flow field, and it is demonstrated that combining the RANS framework with the MRF technique effectively predicts the hydrodynamic coefficients. This approach was adopted due to its significantly reduced computational cost compared to methods with higher fidelity. The governing equations for incompressible Newtonian fluids, and specifically the

continuity and momentum equations, are solved numerically. The SST turbulence model is also implemented, and is coupled with a comprehensive wall treatment strategy to ensure accurate resolution of the near-wall flow. Spatial discretisation is achieved using the finite volume method, with pressure-velocity coupling. The computational setup is designed to capture variations in open water conditions, thereby enabling robust analysis of the hydrodynamic performance.

MESHING TECHNIQUE

A cylindrical computational domain is established around the RDP. The intake is positioned at a distance of $4D$ upstream of the propeller, while the outflow is located $10D$ downstream, as recommended in [32, 22, 33]. In the radial direction, the distance from the RDP to the cylinder surface is $5D$. Based on prior computational experience and the requirement for a boundary layer mesh, an O-XYZ Cartesian coordinate system was used in this investigation, with the centre of the propeller positioned at the origin. The uniform flow at the intake is directed along the positive X-axis, while the Z- and Y-axes form a vertical plane. The computational domain is divided into two regions, a static zone and a rotating zone, with the RDP located in the latter. The rotating zone has a diameter of $1.006D$ and a length of $0.39D$, as illustrated in Fig. 4. A frame of motion with a specific rotational speed relative to the static zone is assigned to the rotating region. To reduce the number of cells involved and hence the computational time, the computational domain is discretised using a hybrid mesh approach. In view of the critical role of the propeller blades in predicting hydrodynamic performance, a high-quality surface mesh is constructed on the blade surfaces. A specialised topology is also employed for the duct, to ensure accurate matching of its profiles.

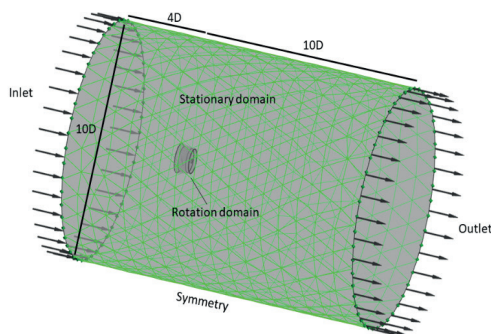


Fig. 4. Computational domains for the analysis of performance and boundary conditions

Prism layers with small growth ratios are applied to all solid surfaces to enhance the resolution of the flow details within the boundary layers. To improve the quality of the mesh in regions with significant curvature, such as near the tip and root of the blade, local refinement is applied. The distribution of on the blade surface is kept below 50, as shown in Fig. 6, and the computational domain contains approximately 12 million elements. The mesh structure near

the wall surface flow gap for the RDPs is illustrated in Fig. 5. The boundary conditions are defined as follows: uniform velocity at the inlet, zero-gradient gauge pressure at the outlet, symmetric conditions on the cylindrical surface, and no-slip conditions at the RDP. A comprehensive summary of the boundary conditions is given in Table 3.

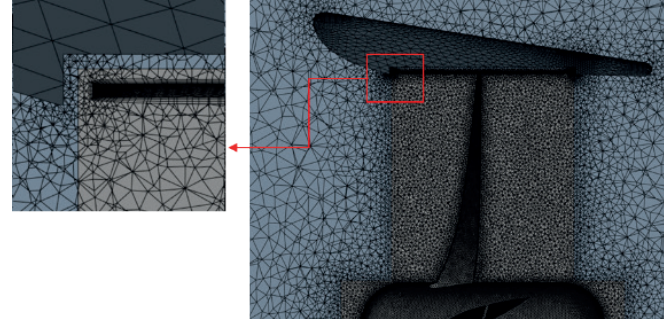


Fig. 5. Volume mesh around the thruster

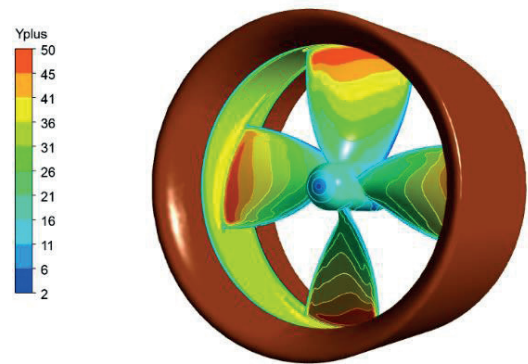


Fig. 6. Y^+ at the wall blades and rotor of the RDP

Tab. 3. Boundary conditions

Patch	Velocity [m/s]	Pressure [pa]	$k \left[\frac{m^2}{s^2} \right]$	$\omega \left[s^{-2} \right]$
Inlet	Based on J	$\frac{\delta p}{\delta n} = 0$	Based on turbulent intensity (1%)	Based on turbulent intensity (1%)
Outlet	$\frac{\delta p}{\delta n} = 0$	P reference	$\frac{\delta k}{\delta n} = 0$	$\frac{\delta \omega}{\delta n} = 0$
Outer boundary	Symmetry	P reference	$\frac{\delta k}{\delta n} = 0$	$\frac{\delta \omega}{\delta n} = 0$
Rotor	No slip, MRF	$\frac{\delta p}{\delta n} = 0$	Wall function	Wall function
Duct	No-slip, fixed	$\frac{\delta p}{\delta n} = 0$	Wall function	Wall function
Blades	No-slip, MRF	$\frac{\delta p}{\delta n} = 0$	Wall function	Wall function

It is essential to verify that the number of mesh elements does not influence the accuracy of the RDP calculation results. The open-water performance of the RDP blade under the operating condition $J=0.1$ was predicted using three different grid sizes, with 5 million, 7 million, and 12 million elements. A comparison of the errors for the data on the RDP blade for these three grid numbers and the results of the simulation calculations are presented in Table 4. An increase in the number of elements led to a reduction in the error, as can be seen from the table. An error analysis indicates that the accuracy of the simulation results improves with a larger number of mesh elements; however, the impact of mesh resolution on the total thrust coefficient and torque coefficient gradually diminishes, and the impact eventually becomes negligible. The finer mesh was chosen for the study due to its ability to capture more detailed flow characteristics while striking a balance between computational efficiency and accuracy.

Tab. 4. Mesh independence for different resolutions at $J=0.1$

Mesh density	Element cells (millions)	K_{tr}	$\Delta K_{tr}\%$	$10K_q$	$\Delta 10K_q\%$
Coarse	5	0.436353	7	0.483847	10
Medium	7	0.451647	4	0.464900	6
Fine	12	0.474295	0.3	0.449349	1

CONVERGENCE STUDY

In the present work, discrete control equation and turbulence models for the RDP are numerically simulated using the finite volume method. Based on the initial design specifications, the propeller's rotational velocity is kept constant at 10 RPS for the initial speed. Seven different operating conditions are considered in the simulation, with variations in the advance coefficient J achieved by altering the inlet velocity. Convergence is defined as the minimum and average residual values falling below 1×10^{-5} . The propulsor thrust and torque values remain stable, and the computation is deemed to have converged after approximately 2250 iterations, representing a good convergence speed.

VALIDATION

The hydrodynamic performance of the DP obtained from the simulation results and experimental data was compared, as shown in Fig. 7. The experimental data were sourced from relevant publications [29]. The thrust coefficient of the duct is notably high at low advance coefficients, suggesting that the duct may be effective in heavy conditions. This would enhance manoeuvrability while maintaining consistent operational conditions. However, as can be seen from the figure, the effect of the duct becomes negative at higher advance coefficients, indicating that the duct may have an adverse effect, reducing thrust under these conditions. The figure shows that the results for the propeller thrust

coefficient (K_{tp}), duct thrust coefficient (K_{td}), and torque coefficient (K_q) of the ducted propeller obtained using CFD are compatible with the experimental data, demonstrating the accuracy of the calculations. It can therefore be concluded that the CFD calculation method employed in this study is capable of accurately predicting the hydrodynamic performance of propellers, and can be effectively used to analyse the hydrodynamic performance of an RDP.

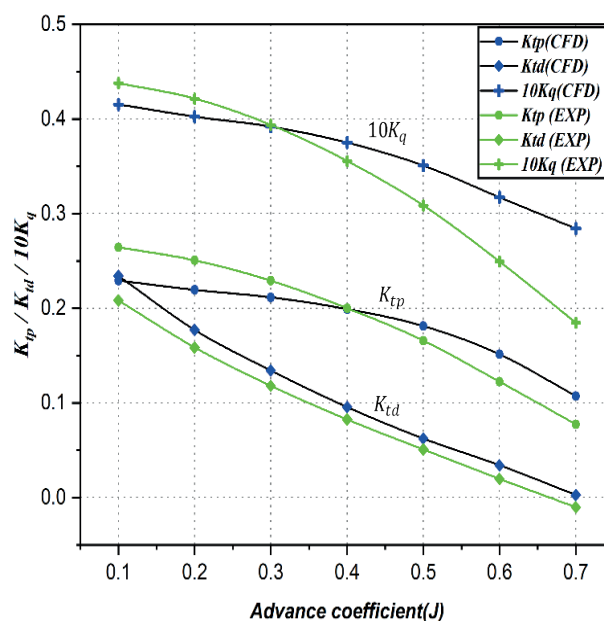


Fig. 7. Comparison of hydrodynamic performance for the DP (experimental data (EXP) from Wang et al. [31])

RESULTS AND DISCUSSION

HYDRODYNAMIC PERFORMANCE

In this section, the hydrodynamic performance of the propellers is presented and compared between the DP and RDP, with a focus on their performance under open water conditions. For all the calculations involving a varying advance coefficient, the rotational velocity of the propeller was maintained at a constant 10 RPS, while the inlet velocity was adjusted accordingly. A Cartesian coordinate system was used, with the X-, Y-, and Z-directions representing the downstream, upward, and starboard directions, respectively. The origin of the coordinate system was positioned at the centre of the propeller hub. The contributions of this study can be summarised as follows:

(1) CFD simulations are conducted to analyse the thrust coefficient, torque coefficient, and efficiency of RDPs with varying advance coefficients. The study also examines the effects of velocity and pressure distribution on the propeller and the modified duct of the RDP, as well as the pressure and velocity flow field in the gap flow area.

(2) The pressure distribution and velocity are analysed at the surface of the blade as well as at different transverse cutting planes downstream of the flow field in the RDP's wake. The TI distributions on the efflux plane and the duct surface are also studied.

The calculation results for the seven operating conditions are presented in Fig. 8, where the red curves show the open water hydrodynamic performance of the RDP, and the green curves represent the corresponding performance of the DP.

The thrust coefficient of the RDP system is significantly higher than for the DP system, where the impeller generates the primary thrust of the propulsor, as shown in Fig. 8. The thrust coefficients of both the DP and RDP systems show a strong linear relationship with the advance coefficient. The thrust coefficient for the DP system is known as resistance when the advance coefficient is greater than 0.5. As shown in Fig. 8, the curves also indicate that as the advance coefficient increases, the torque coefficients and thrust coefficients for the RDP and DP systems show a negative trend. Moreover, the trend in the torque coefficient indicates a correlation between increased RDP torque and rim-driven friction.

Fig. 9 presents a comparison of the open water hydrodynamic efficiency of the RDP and DP systems, based on present numerical CFD results and experimental data. Although the RDP has a lower efficiency than the DP due to the rotating rotor, this does not necessarily indicate that the DP is a superior propulsion system.

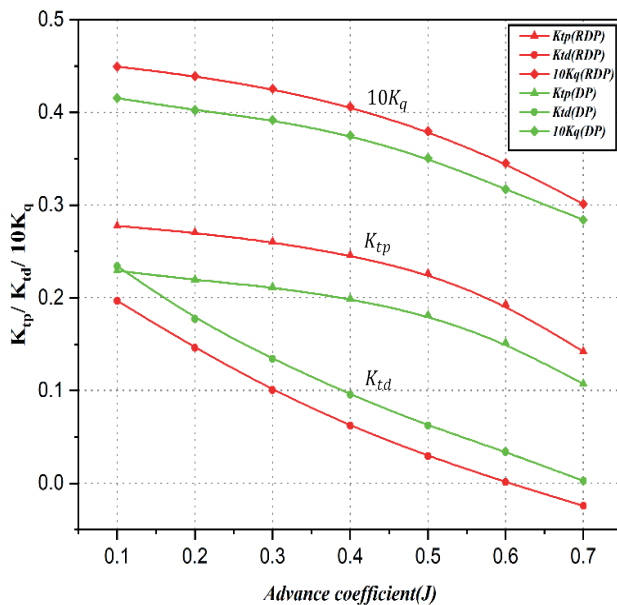


Fig. 9. Comparison of efficiency for the RDP and DP

Fig. 10 shows the relative error in efficiency for the DP and RDP models across a range of advance coefficients (J). The results indicate that the RDP has a slightly higher error due to the rim-driven effect within the duct. The maximum error of approximately 7.9% occurs at $J=0.7$, which falls outside the design scope. For advance coefficients of below 0.7, the average error is approximately 1.62% for the DP and 3.01% for the RDP.

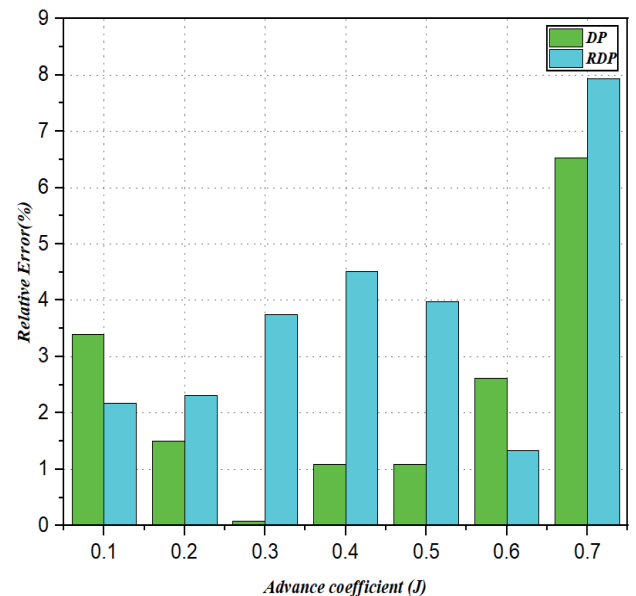


Fig. 10. Relative percentage error in efficiency for the RDP and DP models at different advance coefficients

VELOCITY AND PRESSURE AROUND THE RDP AND DP

Figs. 11 and 12 show the normalised velocity distribution cloud at advance coefficients of $J=0.2$ and $J=0.5$, respectively. The diagram demonstrates that the accelerating flow merges with the incoming flow after passing through the propeller, and convergence is clearly seen in the more distinct hook-shaped vortex formation around the propeller area. This indicates that the RDP accelerates the flow field, which not only improves thrust but also plays a vital role in reducing the energy losses typically associated with drag.

Fig. 13 shows the axial velocity vector field in the blade tip region, with the pressure coefficient (CP) contours serving as the background for both the RDP and DP at $J=0.5$. As the flow approaches the tip clearance area, the boundary layer at the inner surface of the duct begins to separate, due to the

presence of an adverse pressure gradient. This separation occurs because a heavier loading condition results in a higher pressure gradient. At the propeller tip, the flow interacts with the separated boundary layer at the interior surface of the duct, resulting in separation on the pressure side. As the fluid momentum increases, the vortical structures are displaced toward the suction side of the propeller, and diminish in size. In contrast, as shown in Fig. 13(a), the flow field near the blade tip for the RDP is less complex, and backflow develops in the gap due to a pressure drop in that region. The pressure disparity between the pressure and suction sides of the propeller significantly influences the tip leakage vortex, while the induced vortex forms concurrently. The tip separation vortex originates from the separated flow at the propeller tip. These vortical structures interact, leading to viscous losses during their development, which adversely affect the propeller's performance.

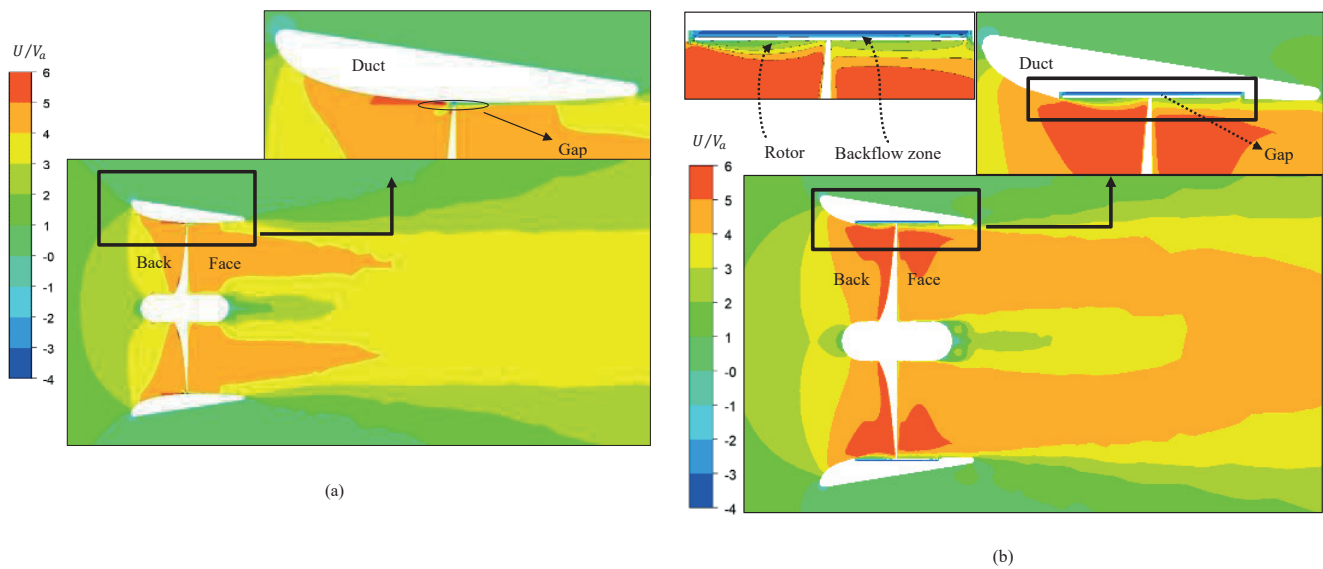


Fig. 11. Contours representing the normalised velocity field around the propeller and duct: (a) DP (b) RDP at $J=0.2$.

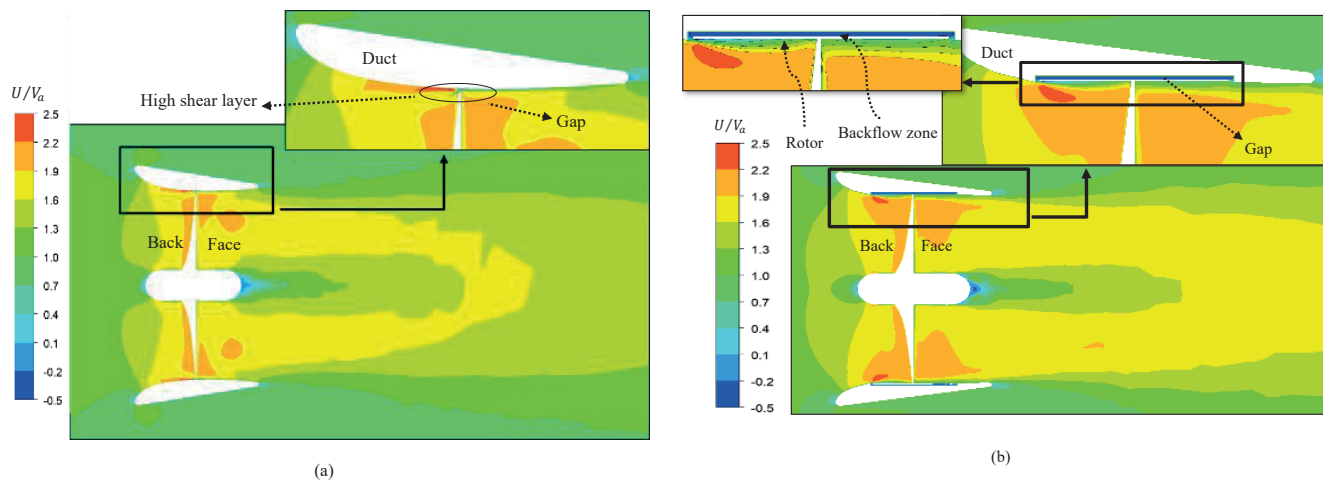


Fig. 12. Contours representing the normalised velocity field around the propeller and duct: (a) DP (b) RDP at $J=0.5$.

PRESSURE COEFFICIENT AT THE BLADE AND DUCT SURFACES

The pressure distribution is essential in order to calculate the propeller performance, as the propeller operates as a lifting body where the pressure forces prevail over the shear stress forces. Figs. 14 and 15 show the pressure coefficient distribution cloud for the impeller at different blade radii (relative radii of 0.3, 0.6, and 0.9 with respect to the section chord) between the RDP and DP at advance coefficients of $J=0.2$ and $J=0.5$, respectively. It is evident that the reduced

pressure around the leading edge of the suction surface may result in cavitation and a decline in propeller performance. The increased pressure differential at the leading edges between the suction and pressure surfaces indicates that negative load is concentrated in this region, leading to negative partial cavitation. To mitigate these effects, it is recommended to reduce the camber, increase the chord length, and modify the leading edge. The remaining regional pressure distributions predominantly exhibit a consistent and uniform pattern, similar to that of airfoils.

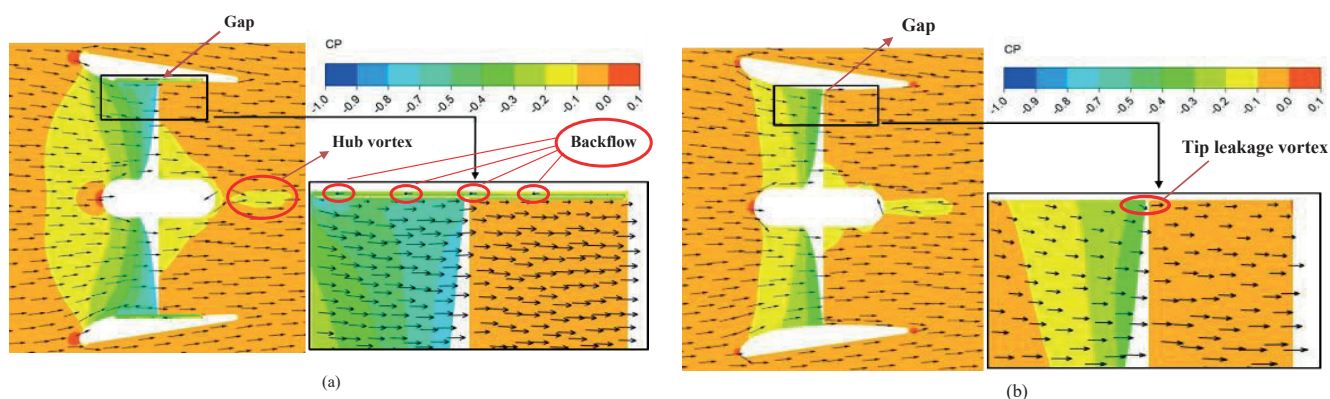


Fig. 13. Contours representing the pressure coefficient with vectors showing the axial velocity fields near the blade tip for: (a) RDP (b) DP at $J=0.5$ (the critical regions of the hub vortex, backflow, and tip leakage vortex are labelled to illustrate the flow interactions)

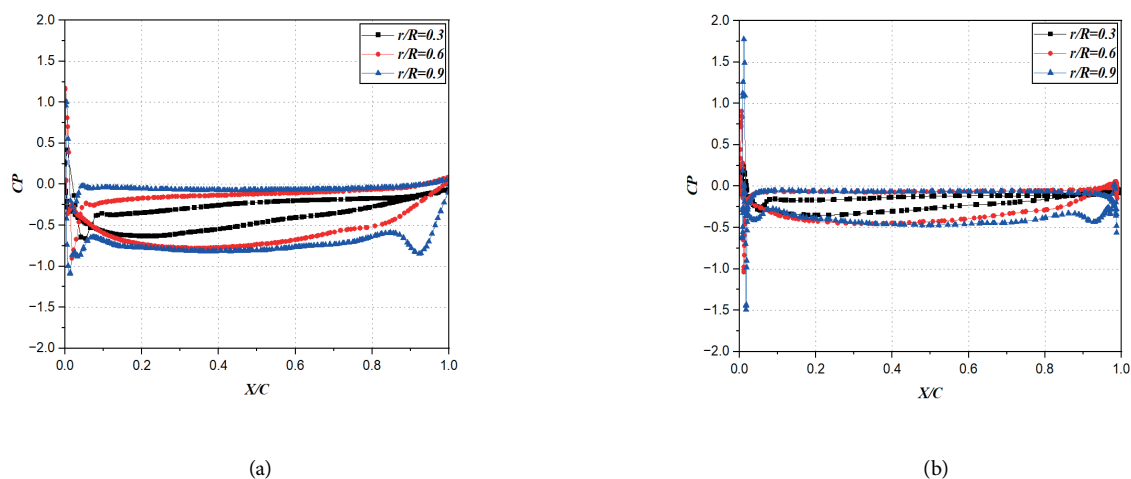
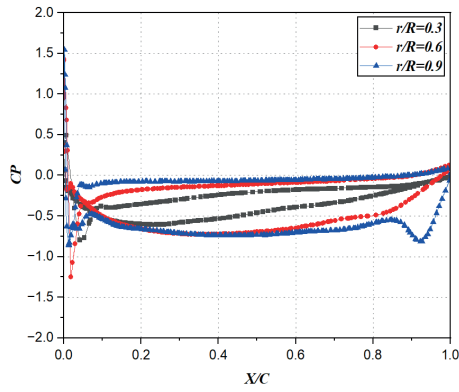
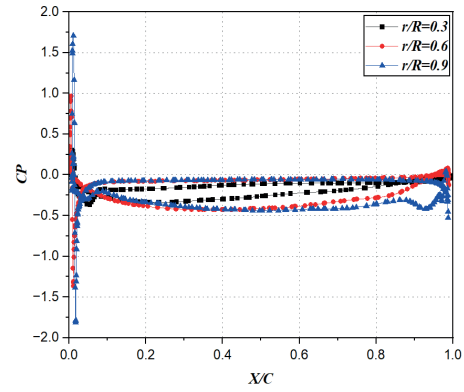


Fig. 14. Pressure coefficients at different radii for (a) RDP and (b) DP at $J=0.2$.

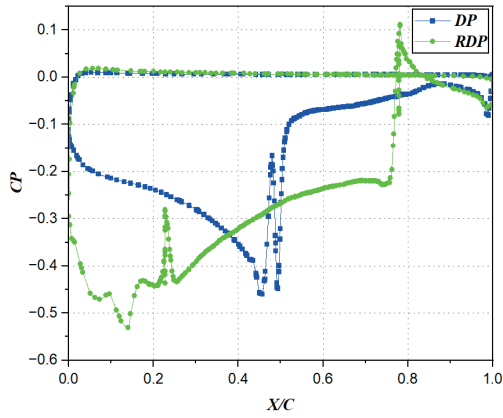


(a)

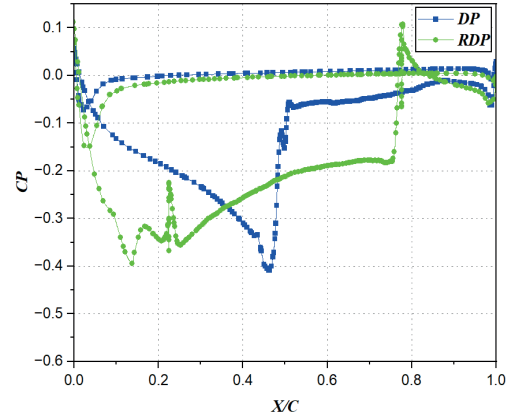


(b)

Fig. 15. Pressure coefficients at different radii for (a) RDP and (b) DP at $J=0.5$.



(a)



(b)

Fig. 16. Pressure coefficients at the duct of the RDP and DP: (a) $J=0.2$; (b) $J=0.5$.

The pressure distribution coefficients for the duct section are depicted in Fig. 16(a) and (b), and the significant impact of the duct shape on the pressure distribution of the RDP can be seen. An optimally designed duct can help to balance the pressure distribution and mitigate cavitation effects, thereby

enhancing performance and efficiency. Fig. 17 shows the pressure coefficient contours at the duct surface for both the RDP and DP. Low-pressure and high-pressure areas at the inner surface can be observed.

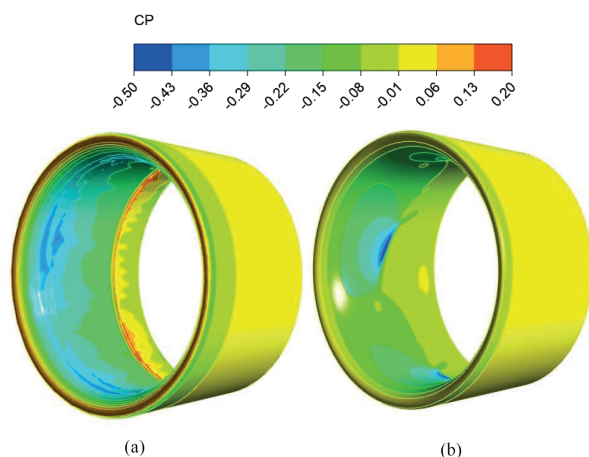


Fig. 17. Pressure coefficient contours at the duct for (a) RDP and (b) DP at $J=0.5$

Fig. 18 presents the pressure distributions at the face and back of the RDP and DP models. The figures show that high pressure is present on the face side, while low pressure is observed at the back. The lowest pressure occurs at the suction surface of the impeller at the blade tip, which is the most likely location for cavitation. The numerical simulation results indicate that the impeller's suction surface for the RDP has a minimum pressure, meaning that cavitation is likely to occur.

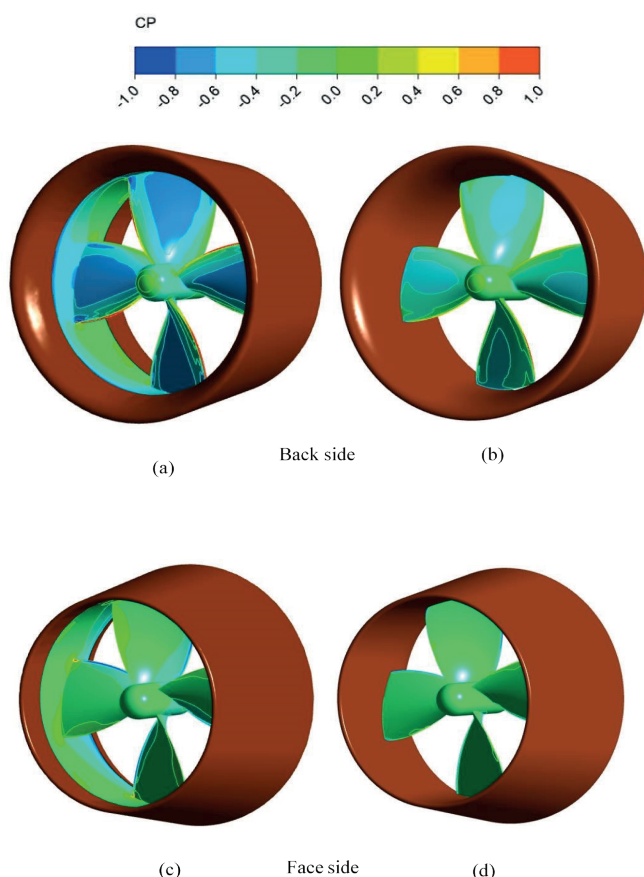


Fig. 18. Pressure coefficient contours on both sides of the blades: (a, c) RDP and (b, d) DP, $J=0.5$

PRESSURE AND VELOCITY CONTOURS AT DIFFERENT DOWNSTREAM POSITIONS

The pressure distributions for the RDP and DP at the different downstream transverse planes are presented in Figs. 19 and 20. The figures clearly indicate that the pressure contour at the downstream position of the propeller for both RDP and DP ($J=0.5$). The presence of a rotor in the system results in a high-pressure region, which is depicted by the red contours.

The aim of this design is to minimise turbulence and optimise the flow path, which is essential in order to maintain the axial velocity. In an RDP, the propeller is mounted on the outer edge of a circular rim. Although this configuration enables a compact design, it also introduces complex flow dynamics. As it operates in open water, the RDP often generates irregular flow patterns, leading to regions of turbulence and fluctuating axial velocities. In contrast, DP configurations are specifically designed to maintain a more uniform and stable axial velocity profile, resulting in higher efficiency and improved performance. The RDP experiences greater variability in axial velocity, which can result in inefficiencies and turbulence, as shown in Figs. 21 and 22. The differences in axial velocity between the DP and RDP arise from their structural designs and flow management characteristics. Notably, the downstream axial velocity is higher for RDP compared to DP.

Cross-flow refers to the movement of water perpendicular to the main flow direction created by the propeller. This phenomenon can significantly impact the performance and efficiency of both RDP and DP, which are enclosed within a duct or shroud, as shown in Figs. 23 and 24. Cross-flow can lead to uneven loading on the propeller blades, resulting in increased drag and reduced efficiency. Cross-flow can also create non-uniform flow conditions, affecting the performance of the propeller and potentially leading to cavitation, and has a significant impact in both the RDP and DP configurations. Downstream cross-flow introduces turbulence and disrupts the intended flow patterns, further impacting performance. Effectively understanding and managing cross-flow are essential in order to optimise the performance and efficiency of these propulsion systems. In DP configurations, downstream cross-flow directly impacts the performance of a propeller by generating turbulence, which interferes with the smooth, laminar flow over the blades, resulting in reduced thrust and increased drag, and ultimately diminishing the overall efficiency. In RDP configurations, downstream cross-flow similarly affects performance. The interaction between the rim and the surrounding fluid creates complex flow patterns, with cross-flow inducing turbulence. This turbulence disrupts the efficiency of the propulsion system, potentially lowering the overall performance and increasing energy consumption. To mitigate the impact of cross-flow, several techniques have emerged, which include optimising the duct shape, adjusting the blade pitch, and implementing flow control devices. However, these modifications may not always result in reduced thrust and torque.

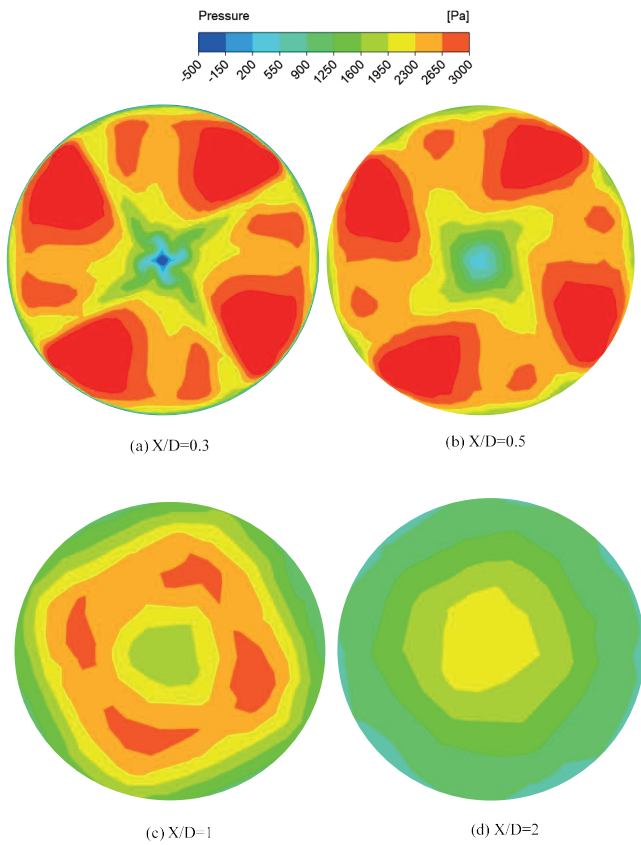


Fig. 19. Pressure contours at different downstream positions for the RDP at $J=0.5$

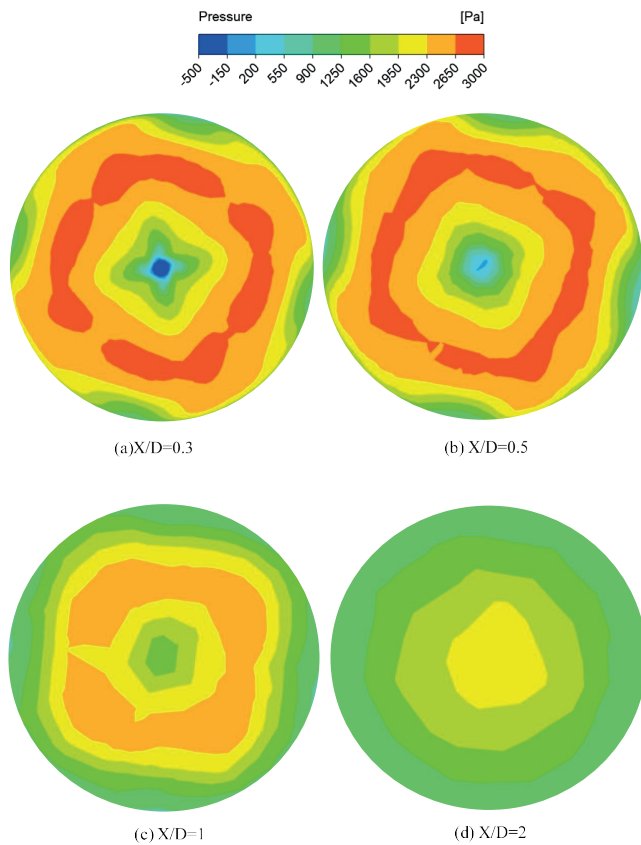


Fig. 20. Pressure contours at different downstream positions for the DP at $J=0.5$

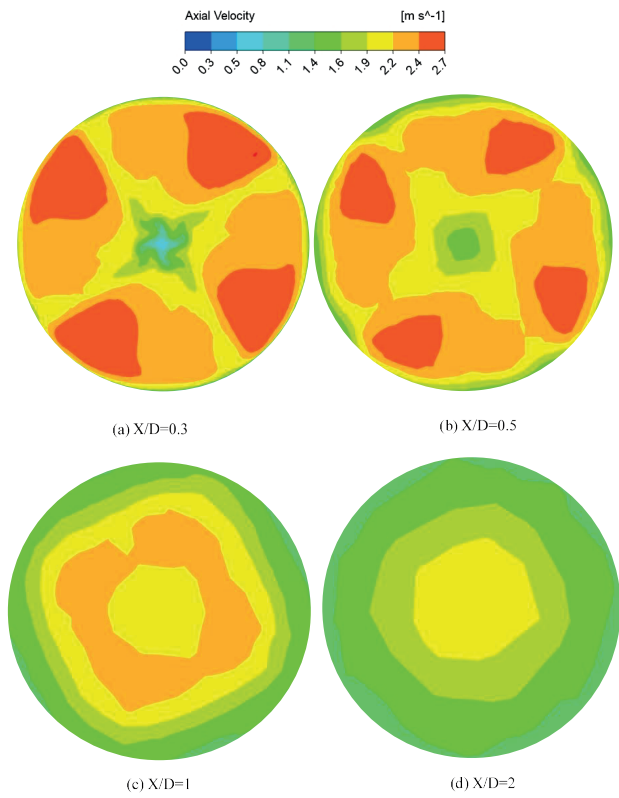


Fig. 21. Axial velocity contours at different downstream positions for the RDP at $J=0.5$

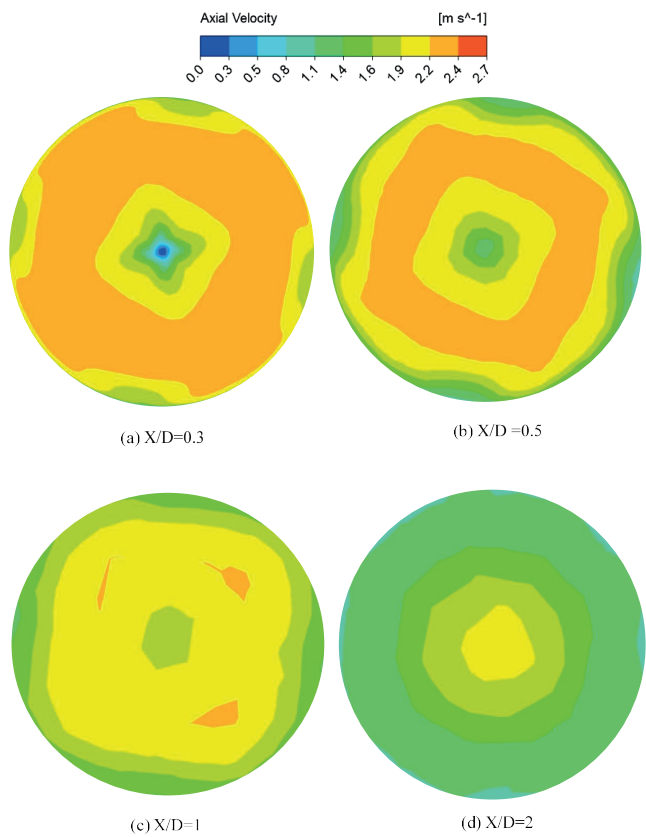


Fig. 22. Axial velocity contours at different downstream positions for the DP at $J=0.5$

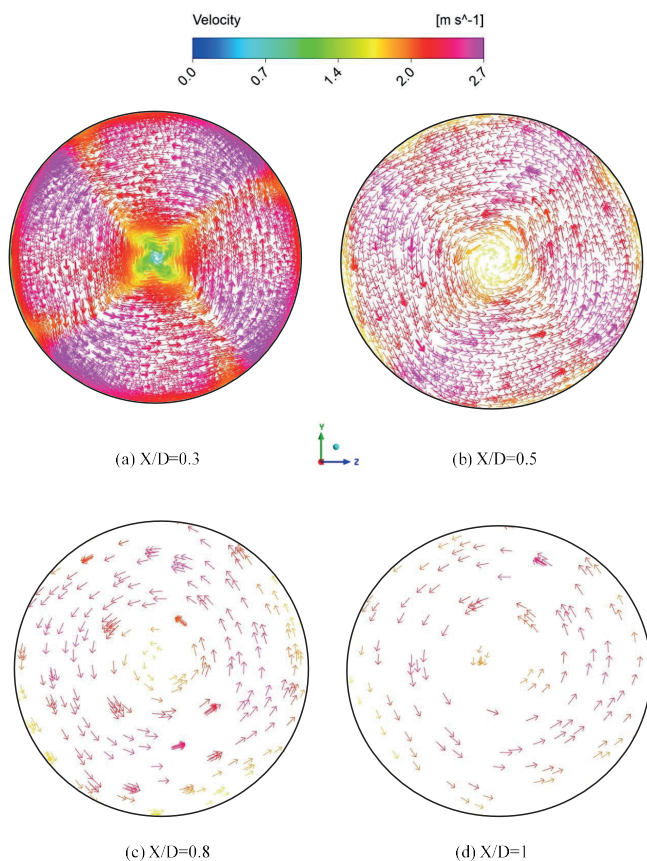


Fig. 23. Cross-flow velocities at different positions for the RDP at $J=0.5$

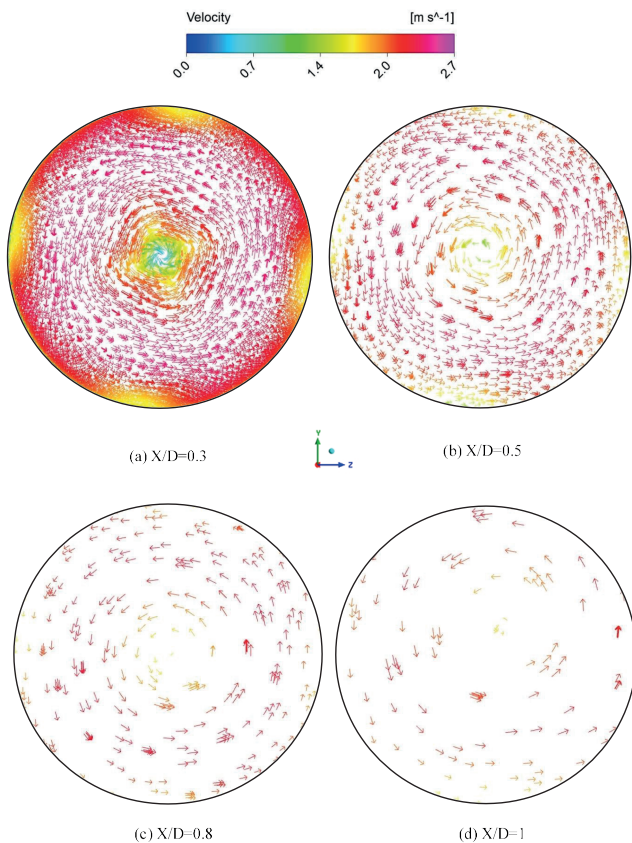


Fig. 24. Cross-flow velocity at different positions for the DP at $J=0.5$

COMPARISON OF THE GAP IN RDP AND DP

In this study, the effects of the pressure coefficient on the gap were examined under different values for the advance coefficient. In the DP, the gap is the spacing between the blade tip and the duct, which plays a critical role in the performance of the DP system, as it facilitates the flow of water while minimising turbulence and drag. It also helps to optimise the flow characteristics, enhance the efficiency, and reduce the risk of cavitation. In the RDP system, the gap refers to the clearance between the rotor and the duct. This gap is important for the interaction between the rotor and the water, as it allows water to flow into the rotor while enhancing the backflow in the gap areas. The pressure coefficients between points A and B within the gap areas of both the RDP and DP were evaluated as shown in Fig. 25.

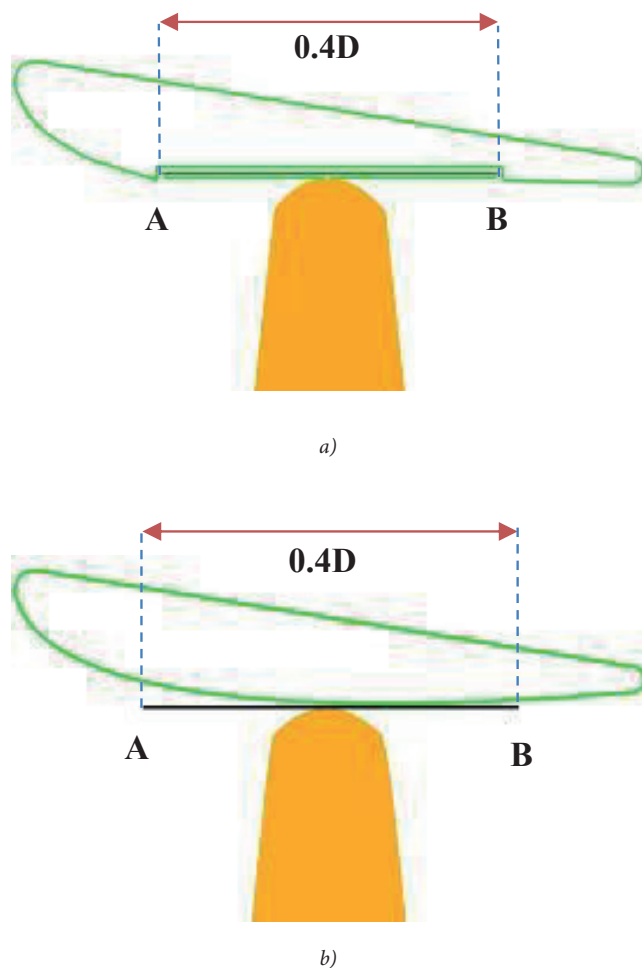


Fig. 25. Layout of the gap for (a) RDP and (b) DP

Both the DP and RDP rely on this gap for optimal performance, although the specifics of their design and their impact on performance differ due to variations in their structural configuration. An understanding of these differences is essential in order to optimise each type of propulsion system. The values of the pressure coefficient

in the gap between points A and B for both the RDP and DP are shown in Fig. 26. The pressure coefficients for the DP and RDP differ significantly: the DP relies on a pressure

drop across the blades to generate thrust, while the RDP uses a low-pressure zone created by rotor motion, which can lead to more effective pressure recovery.

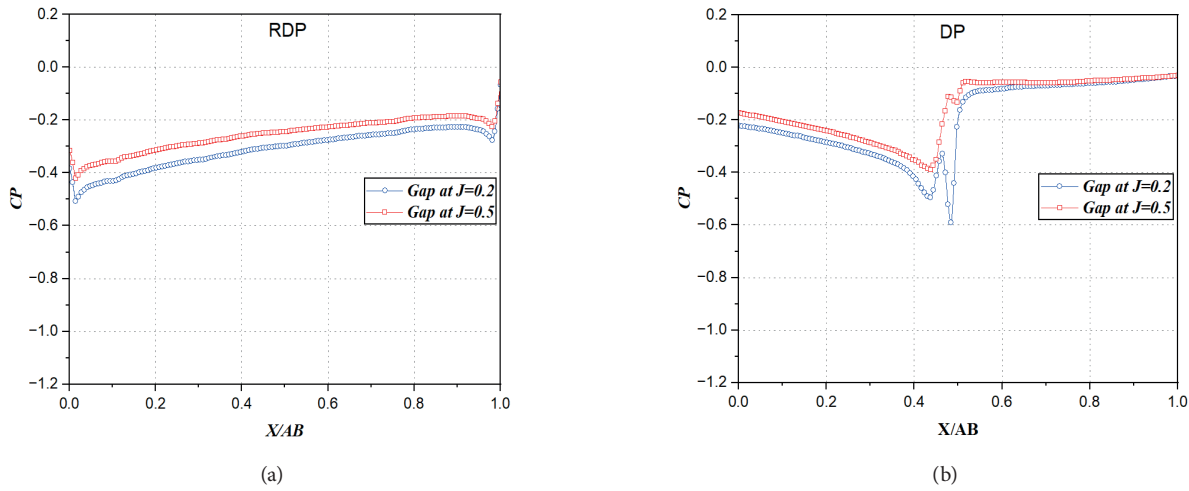


Fig. 26. Pressure coefficient in middle of the gap for the (a) RDP and (b) DP at different advance coefficients

The rotor in an RDP plays an important role in its operation and efficiency. Positioning the rotor at the outer edge enhances the thrust generation, reduces the structural complexity, and improves the overall performance, making the RDP a valuable form of technology for modern marine propulsion systems. Fig. 27 shows the values of the pressure coefficient around the rotor on the RDP at different advance coefficients ($J=0.2$ and $J=0.5$), providing essential insights into the thrust generation and thruster efficiency. The pressure coefficient at the rotor of an RDP is an essential parameter when evaluating and optimising the performance of these propulsion systems. An analysis of the CP can lead to improvements in the design, efficiency, and effectiveness of thrusters for various applications.

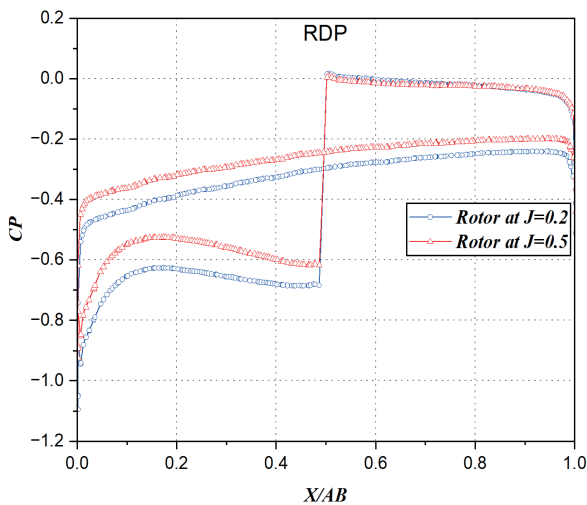


Fig. 27. Results for the pressure coefficient at the rotor of the RDP at different advance coefficients

TURBULENCE INTENSITY

The rotating propeller induces periodic hydrodynamic forces in the surrounding water, and this interaction between the propeller and the initially static fluid generates localised flow disturbances. TI can be used as a metric to quantify the velocity fluctuations caused by these hydrodynamic irregularities. For each velocity component, TI depends on the turbulent kinetic energy (k) divided by the mean flow velocity (U_{avg}), and is calculated using Eq. (10):

$$\text{Turbulence intensity} \quad TI = \frac{\sqrt{\frac{2}{3} K}}{U_{avg}} \quad (10)$$

Fig. 28 shows the TI contour distributions for both the DP and RDP at advance coefficients of $J=0.2$ and $J=0.5$, respectively. There is a 20% reduction in turbulence for the RDP at $J=0.5$, as can be seen from Fig. 28, which makes it suitable for stealth-critical missions, despite an 8% decrease in thrust compared to the DP. Although the RDP has a higher TI near the rotor-duct gap, this broadband turbulence is less acoustically efficient than the concentrated tip vortices characteristic of DPs, which are known to generate significant noise. These findings are particularly relevant to the deployment of RDP in AUVs and bow thrusters, where compactness and noise reduction are prioritised over moderate efficiency losses. The TI results differ between the DP and RDP configurations, as illustrated in Fig. 29, which shows the TI percentages at the inner and outer surfaces of the duct for the DP and RDP at $J=0.2$ and $J=0.5$, respectively. For the RDP, the maximum TI at $J=0.2$ was 55%, while at

$J=0.5$, it decreased to approximately 22%. As the advance coefficient increases, the TI value shows a corresponding decrease, indicating a reduction in flow irregularities. The

TI distribution for each propeller exhibits peak values, with the highest peaks occurring at the blade tip position for the DP and the rotor position for the RDP.

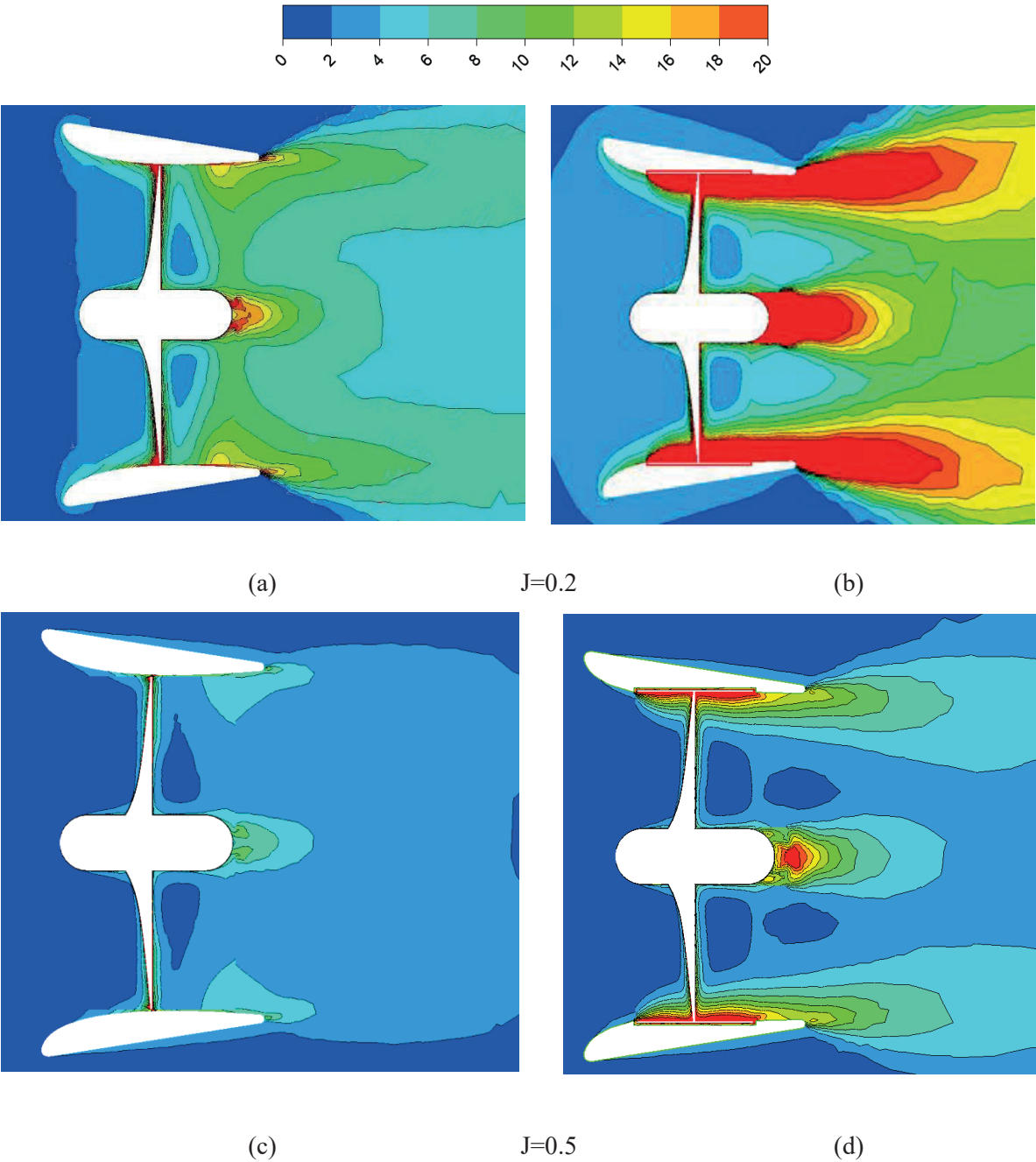


Fig. 28. TI contour distributions for (a, c) DP and (b, d) RDP at $J=0.2$ and $J=0.5$

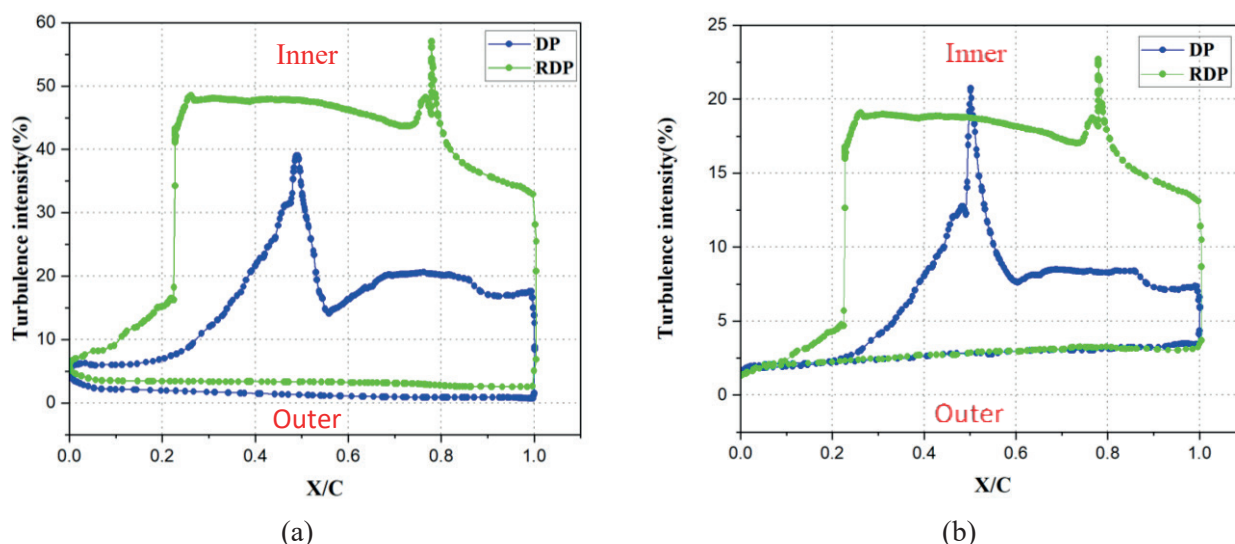


Fig. 29. TI percentage on the Inner and Outer surfaces of the duct for DP and RDP at (a) $J=0.2$ and (b) $J=0.5$.

CONCLUSIONS

This study has investigated the hydrodynamic performance of both a DP and RDP through numerical simulations. The geometries of the DP and RDP were identical except for the addition of a rotor in the RDP configuration. Numerical simulations were carried out using the RANS equations with the SST turbulence model. A Ka-series propeller and a modified +19A duct were selected for the analysis, and numerical results for the hydrodynamic performance were obtained. Based on these results, the following conclusions can be drawn:

- For the DP, the hydrodynamic performance obtained from the simulations was in good agreement with the experimental data. The effect of the rim in the RDP caused a decrease in the performance of the duct thrust due to the gap at the rim.
- At small advance coefficients, the thrust coefficient of the duct is high for both the DP and RDP.
- The efficiency of the RDP is lower than for the DP due to the rim-driven design inside the duct.
- The pressure coefficient distribution at the inside surface of the duct for the RDP is relatively moderate and uniform, except for a positive jump in pressure at the face side of the blade, while for the DP, there is a negative jump pressure at the back of the blade.
- The pressure distribution coefficients at the back of the blade and near the tip decreased by up to for RDR at $J=0.2$.
- Axial and cross-flow velocity and pressure contours at different locations downstream of the DP and RDP have been presented and discussed.
- The values of the TI are different for the DP and RDP. RDP has the highest TI, since the rotor is inside the duct.

Future work will include:

- Greater emphasis on and further analysis of the gap for RDP.
- A new duct design for the RDP to increase the efficiency and performance.

REFERENCES

1. Yan X, Liang X, Ouyang W, Liu Z, Liu B, Lan J. A review of progress and applications of ship shaft-less rim-driven thrusters. *Ocean Engineering* 2017, vol. 144, pp. 142–56. <https://doi.org/10.1016/j.oceaneng.2017.08.045>
2. Wu Z, Gong J, Ding J, Sun Y, Ma C. Autonomous modification and optimization method for rim-driven system in surface ships. *Ocean Engineering* 2023, vol. 290, p. 116293. <https://doi.org/10.1016/j.oceaneng.2023.116293>
3. Gaggero S. A study on the wake evolution of a set of RIM-driven thrusters. *Journal of Marine Science and Engineering* 2023, vol. 11, no. 9, p. 1659. <https://doi.org/10.3390/jmse11091659>
4. Gong J, Ding J, Wang L. Propeller–duct interaction on the wake dynamics of a ducted propeller. *Physics of Fluids* 2021, vol. 33, no. 7. <https://doi.org/10.1063/5.0056383>
5. Razaghian AH, Ghassemi H. Numerical analysis of the hydrodynamic characteristics of the accelerating and decelerating ducted propeller. *Zeszyty Naukowe Akademii Morskiej w Szczecinie*. 2016, vol. 47, no. 119, pp. 42–53. <https://doi.org/10.17402/147>

6. Jiang H, Wu H, Chen W, Zhou P, Zhong S, Zhang X, Zhou G, Chen B. Toward high-efficiency low-noise propellers: A numerical and experimental study. *Physics of Fluids* 2022, vol. 34, no. 7. <https://doi.org/10.1063/5.0098891>
7. Kort L. Elektrisch angetriebene schiffsschraube. German Patent: DE688114, 1940.
8. Lu NX, Bensow RE, Bark G. Large eddy simulation of cavitation development on highly skewed propellers. *Journal of Marine Science and Technology* 2014, vol. 19, pp. 197–214. <https://doi.org/10.1007/s00773-013-0240-3>
9. Phillips AB, Turnock SR, Furlong M. Evaluation of manoeuvring coefficients of a self-propelled ship using a blade element momentum propeller model coupled to a Reynolds averaged Navier Stokes flow solver. *Ocean Engineering* 2009, vol. 36, nos. 15–16, p. 1217–25. <https://doi.org/10.1016/j.oceaneng.2009.07.019>
10. Lin J, et al. Hydrodynamic performance of a rim-driven thruster improved with gap geometry adjustment. *Engineering Applications of Computational Fluid Mechanics* 2023, vol. 17, no. 1, p. 2183902. <https://doi.org/10.1080/19942060.2023.2183902>
11. Wei X, Yan T, Sun T, Liu S, Du H. Research on the hydrodynamic performance of propellers under oblique flow conditions. *Proceedings of the Institution of Mechanical Engineers, Part M: Journal of Engineering for the Maritime Environment* 2024, 14750902241231349. <https://doi.org/10.1177/14750902241231349>
12. Gaggero S, Vernengo G, Villa D, Bonfiglio L. A reduced order approach for optimal design of efficient marine propellers. *Ships and Offshore Structures* 2020, vol. 15, no. 2, pp. 200–214. <https://doi.org/10.1080/17445302.2019.1606877>
13. Hassannia A, Darabi A. (2013). Design and performance analysis of superconducting rim-driven synchronous motors for marine propulsion. *IEEE Transactions on Applied Superconductivity*, 24(1), 40–46. <https://doi.org/10.1109/TASC.2013.2280346>
14. Liang J, Zhang X, Qiao M, Zhu P, Cai W, Xia Y, Li G. Optimal design and multifield coupling analysis of propelling motor used in a novel integrated motor propeller. *IEEE Transactions on Magnetics* 2013, vol. 49, no. 12, pp. 5742–5748. DOI: 10.1109/TMAG.2013.2241776
15. Li C, Liu N, Su J, Hua H. Vibro-acoustic responses of a coupled propeller-shaft-hull system due to propeller forces. *Ocean Engineering* 2019, vol. 173, pp. 460–8. <https://doi.org/10.1016/j.oceaneng.2018.12.077>
16. Sharkh SM, Lai SH. Slotless PM brushless motor with helical edge-wound laminations. *IEEE Trans. Energy Convers.* 2009, vol. 24, pp. 594–598. <https://doi.org/10.1109/TEC.2009.2025423>
17. Matuszewski L. Ring thruster—A preliminary optimisation study. *Polish Maritime Research* 2009, 1(59), vol. 16. pp. 43–46. <https://doi.org/10.2478/v10012-008-0009-5>
18. Cao QM, Hong FW, Tang DH, Hu FL, Lu LZ. Prediction of loading distribution and hydrodynamic measurements for propeller blades in a rim driven thruster. *Journal of Hydrodynamics* 2012, vol. 24, no. 1, pp. 50–7. [https://doi.org/10.1016/S1001-6058\(11\)60218-7](https://doi.org/10.1016/S1001-6058(11)60218-7)
19. Song S, Demirel YK, Atlar M. Penalty of hull and propeller fouling on ship self-propulsion performance. *Applied Ocean Research* 2020, vol. 94, p. 102006. <https://doi.org/10.1016/j.apor.2019.102006>
20. Cai B, Mao X, Xu Q, Chai W, Tian B, Qiu L. Simulation of the interaction between ship and ducted propeller with a modified body force method. *Ocean Engineering* 2022, vol. 249, p. 110950. <https://doi.org/10.1016/j.oceaneng.2022.110950>
21. Menter FR. Two-equation eddy-viscosity turbulence models for engineering applications. *AIAA J.* 1994, vol. 32, pp. 1598–1605. <https://doi.org/10.2514/3.12149>
22. Dubas AJ, Bressloff NW, Sharkh SM. Numerical modelling of rotor–stator interaction in rim driven thrusters. *Ocean Engineering* 2015, vol. 106, pp. 281–8. <https://doi.org/10.1016/j.oceaneng.2015.07.012>
23. Majdfar S, Ghassemi H, Forouzan H, Ashrafi A. Hydrodynamic prediction of the ducted propeller by CFD solver. *Journal of Marine Science and Technology* 2017, vol. 25, no. 3, p. 3. <https://doi.org/10.6119/JMST-016-1214-2>
24. Feng D, Yu J, He R, Zhang Z, Wang X. Improved body force propulsion model for ship propeller simulation. *Applied Ocean Research* 2020, vol. 104, p. 102328. <https://doi.org/10.1016/j.apor.2020.102328>
25. Hu J, Li T, Guo C. Two-dimensional simulation of the hydrodynamic performance of a cycloidal propeller. *Ocean Engineering* 2020, vol. 217, p. 107819. <https://doi.org/10.1016/j.oceaneng.2020.107819>
26. Zhang S, Zhu X, Zhou ZL. Hydrodynamic performance analysis of hubless rim-driven propulsors. *Applied Mechanics and Materials* 2013, vol. 256, pp. 2565–8. <https://doi.org/10.4028/www.scientific.net/AMM.256-259.2565>
27. Zhu Z, Liu H. The external characteristics and inner flow research of rim-driven thruster. *Advances in*

Mechanical Engineering 2022, vol. 14, no. 2. <https://doi.org/10.1177/16878132221081608>

28. Koronowicz T, Krzemianowski Z, Tuszkowska T, Szantyr JA. A complete design of ducted propellers, using the new computer system. Polish Maritime Research 2(60) 2009, vol. 16, pp. 34–39. <https://doi.org/10.2478/v10012-008-0019-3>
29. Majdfar S, Ghassemi H, Forouzan H, Ashrafi A. Hydrodynamic prediction of the ducted propeller by CFD solver. Journal of Marine Science and Technology 2017, vol. 25, no. 3, p. 3. <https://doi.org/10.6119/JMST-016-1214-2>
30. Yang C, et al. Numerical study of relationships between flows and structural characteristics of the rotor in a rim-driven hubless thruster using a strongly-coupling FSI algorithm. Ocean Engineering 2025, vol. 323. <https://doi.org/10.1016/j.oceaneng.2025.120560>
31. Wang KC, Liu HY, Tsao ML, Chu HH. Ducted propellers with simplified duct profile. J. Shipbuild. China 1978, vol. 9, p. 63. <https://trid.trb.org/View/148019>
32. Liu B, Vanierschot M. Numerical study of the hydrodynamic characteristics comparison between a ducted propeller and a rim-driven thruster. Applied Sciences 2021, vol. 11, no. 11, p. 4919. <https://doi.org/10.3390/app11114919>
33. Song BW, Wang YJ, Tian WL. Open water performance comparison between hub-type and hubless rim driven thrusters based on CFD method. Ocean Engineering 2015, vol. 103, pp. 55–63. <https://doi.org/10.1016/j.oceaneng.2015.04.074>