DOI: 10.15480/882.9331

Eighth International Symposium on Marine Propulsors
smp'24, Berlin, Germany, March 2024

A partial accelerating duct over the marine propeller

Anirban Bhattacharyya[1], Kiran Ramesh[1]

[1]Department of Ocean Engineering and Naval Architecture, Indian Institute of Technology Kharagpur, West Bengal, India

ABSTRACT

The hydrodynamic performance of a marine propeller in combination with a partial accelerating duct is investigated. The duct is placed over the upper half of the propeller such that its distance from the blade tip is minimum in the vertical position (0 deg), and gradually increases to a maximum value for the horizontal blade positions (90 deg). The elliptical geometry allows a gradual transition for the propeller blades from the 'ducted condition' in the upper half to the 'open condition' in the lower half over one propeller revolution. CFD investigations are performed at two scales to understand the influence of the duct on the open water characteristics. The effect of duct geometry is studied by varying the axis ratio. Model self-propulsion show 2-3% improvement in the propeller thrust/torque ratio at higher propeller loadings with the partial duct.

Keywords

Marine Propeller; Partial Duct; Open Water Characteristics; Pressure Coefficient.

1 INTRODUCTION

It has been well established that once the propeller design is optimized for a specific vessel, energy losses can be reduced by placing suitable devices upstream (pre-swirl) or downstream (post-swirl) of the propeller. The strict norms for energy efficiency implemented by the International Maritime Organization (IMO) have led to a surge of energy saving devices (ESD) being installed in ships over the past decades. Based on the hydrodynamic mechanisms, different types of ESD have been designed and effectively employed on ocean going vessels. While the pre-swirl devices primarily aim the reduction of non-uniformities in the inflow, the post-swirl devices are generally used to reduce the energy losses in the propeller slipstream. A general overview of the hydrodynamic mechanisms of these installations can be found in (Terwisga, 2013). Some of the most widely used pre-swirl duct designs are: Wake Equalising Duct (Schneekluth 1986), Sumitomo Integrated Lammeren Duct (Sasaki and Aono 1997), and Mewis Duct (Mewis 2009).

The present work investigates the propeller thrust, torque, and flow hydrodynamics when used in conjunction with a partial flow accelerating duct (Bhattacharyya 2020). The idea behind such a device is that the flow acceleration due to the duct will be only in the upper sections of the propeller plane, where the wake fractions due to the effect of the ship stern are higher. Unlike typical stern-mounted partial ducts (Korkut 2006) placed ahead of the propeller, the partial duct adopted in the present study is located around the propeller. Hence, it is considered as a part of the propulsion system, similar to a ducted propeller. Open water performance of the propeller and duct system is investigated using CFD to evaluate the effect of the duct on the thrust, torque and efficiency at varying advance coefficients. An effort has been made to look into important aspects like flow features, pressure distributions and Reynolds number effects. Finally, some results of model self-propulsion tests in the towing tank are presented to provide a simplistic estimate of the relative performance in ducted and open conditions.

2 DUCT GEOMETRY

The partial duct is configured in the form of a semi-ellipse having the sectional geometry similar to that of the Wageningen 19A duct. A representative diagram of the partial duct around a propeller is shown in Fig. 1. The ratio of the horizontal (major axis) and vertical (minor axis) dimensions from the propeller shaft axis to the inner surface of the semi-elliptic duct is taken as 1.5.

Fig. 1: Partial duct around the marine propeller

The influence of this design ratio on the forces and flow characteristics in open water is also investigated in the present work. The two free ends of the duct are faired in order to reduce the strength of tip vortices and the resulting drag. A 5-bladed B-series propeller with expanded blade area ratio of 0.75 has been selected for the present study.

3 INVESTIGATIONS

The primary focus of the present work is to understand the open water performance of a propeller when it operates within a partial duct. Though the investigations pertain to a specific accelerating duct design over the propeller, a similar situation may arise for specific inland vessels with stern tunnel designs, where the propeller operates in partial enclosed conditions.

Hence, considering the duct as a part of the propulsor, the open water characteristics of the propeller and duct system can be expressed as the variation of the propeller thrust coefficient (K_{TP}), the duct thrust coefficient (K_{TD}), the torque coefficient (K_Q), and the open water efficiency (η_O) as a function of the advance coefficient (J). They are expressed as below where 'D' is the propeller diameter and 'n' is the revolution speed.

$$K_{TP} = \frac{T_P}{\rho n^2 D^4} \tag{1}$$

$$K_{TD} = \frac{T_D}{\rho n^2 D^4} \tag{2}$$

$$K_Q = \frac{Q}{\rho n^2 D^5} \tag{3}$$

$$\eta_O = \frac{K_{TP}+K_{TD}}{K_Q} \frac{J}{2\pi} \tag{4}$$

3.1 CFD Methodology

The numerical investigations for the propeller with and without duct in open water are performed using the RANS solver of the commercial software Star-CCM+. A cylindrical computational domain extending to 5D upstream and 11D downstream of the propeller is used. The relative motion between the rotating propeller and the stationary partial duct is considered using appropriate wall boundary conditions. The k-ω SST turbulence model is used along with a coupled implicit solver. The rotational speed is fixed at 12 rps and inlet velocity is changed accordingly based on the required range of the advance coefficient (J). The mesh is composed of polyhedral elements along with prism layers over the propeller and duct surface to capture the boundary layer velocities and flow characteristics. A sectional view of the core mesh around the propeller and the partial duct is shown in Fig. 2.

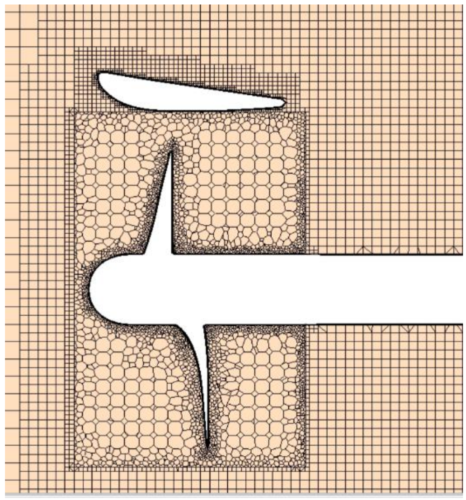

Fig. 2: Sectional view of mesh around propeller and duct

A grid convergence study has been undertaken using four different grids with successive refinement by a factor of √2, as per standard norms (ITTC 2008). The convergence of propeller thrust and torque coefficients with varying mesh size for J=0.2 is shown in Fig. 3. Based on the uncertainty analysis, the chosen grid (Fine 1) has a calculated value of estimated error less than 2% based on the convergence of thrust and torque coefficients.

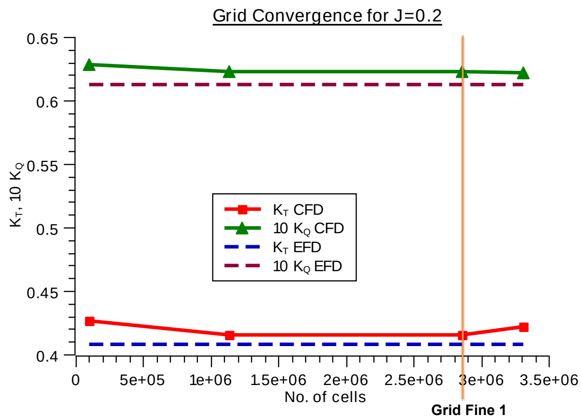

Fig. 3: Results of grid convergence study

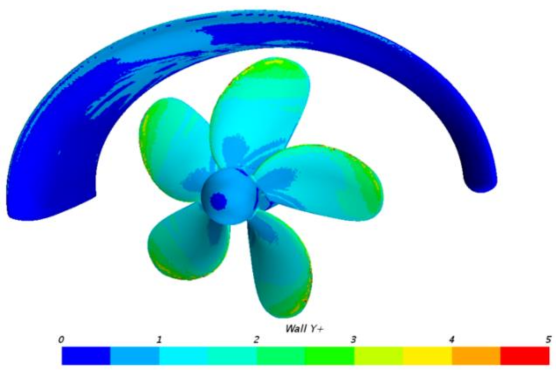

Fig. 4: Wall y+ on the propeller blade and duct (model scale) for the selected grid.

The selected mesh has an average value of the dimensionless wall distance of the first cell y+ < 2 over the propeller and partial duct in the model scale, as shown in Fig. 4. A stretching factor of 1.2 is used for the prism layers. Both steady Moving Reference Frame (MRF) and unsteady approaches have been used. The computations have been undertaken over a range of J from 0.1 to 0.7. Comparison with published open water characteristics of the B 5-75 propeller (Fig. 5) is acceptable (differences of 1-3% for J=0.1 to 0.4) considering the range of interest of propeller loading in the present study.

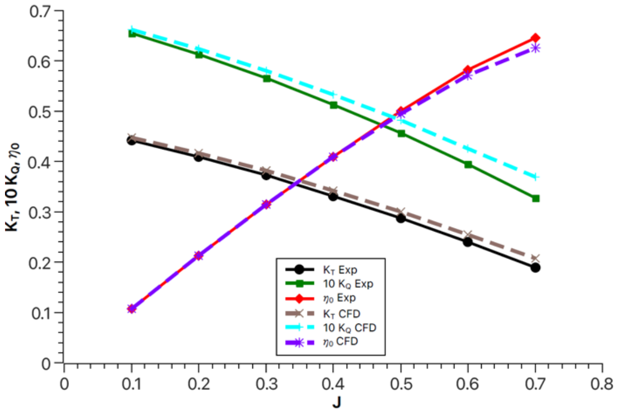

Fig. 5: Validation of open water characteristics obtained from CFD with published experiments. (B5-75 propeller)

The primary CFD study is performed on the model scale for a 0.25m diameter propeller having a chord-based Reynolds number in the range of 10^5. A brief study of the scale effect on open water characteristics is also done which is consistent with the investigations of ducted propeller presented in (Bhattacharyya et al. 2016). The propeller and duct dimensions are scaled up 20 times that of the model values, the propeller speed of advance and the rotational speed being scaled based on Froude identity. The prism layer mesh is adjusted to obtain the target y+ range while the volumetric mesh strategy and local refinement are similar in both scales.

3.2 Model Tests

A set of initial experiments consisting of model propulsion tests are conducted in the 150m long towing tank at the Department of Ocean Engineering and Naval Architecture at IIT Kharagpur. A twin-screw vessel model is fitted alternately with and without the partial duct. The setup for load varying self-propulsion test is used where the force along the direction of ship motion is measured by the resistance dynamometer, and the propeller thrust and torque are measured by a propeller dynamometer for a fixed carriage speed based on Froude number identity. For a specific towing speed, three values of propeller rpm are used to get the model self-propulsion point.

4 RESULTS AND DISCUSSION

The computational study aims to investigate the open water performance of the propeller and the partial accelerating duct combination. It must be mentioned here that any additional device which is placed around the ship stern ahead or astern of the marine propeller will increase the total vessel resistance due to its own drag, However, depending on the geometric configuration and placement with respect to the propeller, such devices may be effectively used to reduce energy losses on the merit of certain hydrodynamic advantages. Hence, the net effect after accounting of the increased drag should be a reduction in the powering requirement. In the present work, a novel partial duct is designed with similar intentions.

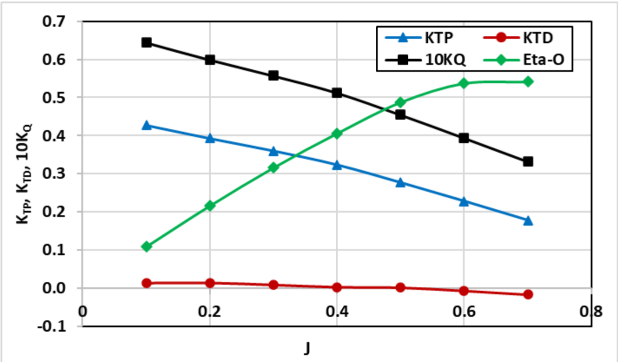

Fig. 6: Open water characteristics of B5-75 propeller within the partial duct.

The thrust generated by the partial duct in open water is found to be quite insignificant as observed in the open water diagram (Fig. 6). The duct thrust is only 3.5% of the propeller thrust at J=0.2, and decreases steadily at higher J values. Compared to simulations without the duct it is observed that the thrust and torque for a particular advance coefficient are reduced due to the flow acceleration by the partial duct which results in a change of the open water efficiency (Fig. 7) depending on their relative variation and the contribution from the duct itself.

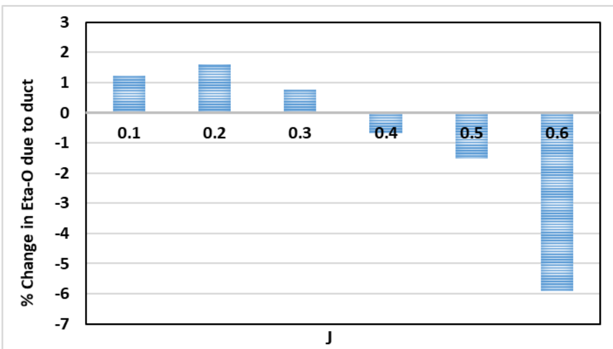

Fig. 7: Effect of the partial duct on the open water efficiency expressed as the percentage change from non-ducted condition.

It is observed that the small improvement in the open water efficiency is restricted to high propeller loading conditions. However, due to its partial design, the net thrust generated is very small as compared to the standard annular 19A duct. This may be attributed to the flow around the elliptical geometry as well as the higher drag due to the end vortices.

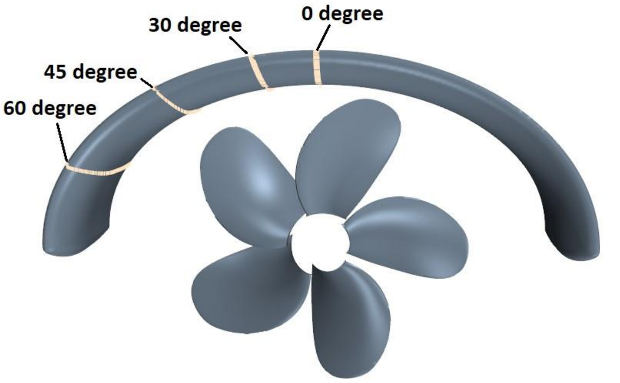

Fig. 8: Four reference sections at 0°, 30°, 45° and 60° on the partial duct.

To investigate this aspect, the pressure distribution over different duct sections (shown in Fig. 8) are compared. At the vertical section (0 deg), the propeller blade is passing closest to the duct, and the distance increases with the increase of the angle from the vertical axis. The pressure coefficient plots shown in Fig. 9 indicate that the sections at 0 deg and 30 deg have greater differences in the pressure coefficient (C_P) between the inner and outer surfaces of the duct. At the outer sections, the distance of the duct from the blade tip increases and the difference in C_P decreases resulting in much lower net thrust.

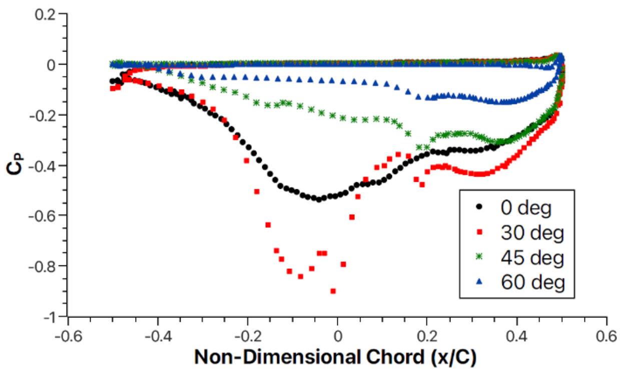

Fig. 9: Pressure coefficient over different sections of the partial duct in the model scale at J=0.2.

The rpm-based velocity component 'nD' is used to compute C_P. The position is non-dimensionalized using the duct chord length and the leading and trailing edges are located at -0.5 and 0.5 respectively. Similarly, the pressure coefficient over a representative propeller blade section at 0.7R is plotted in Fig. 10 in presence of the partial duct and compared with the non-ducted condition.

At a relatively high propeller loading (J=0.2), the effect of the duct is visualized in reduced pressure difference between the pressure and suction sides of the propeller blade resulting in a decrease of propeller thrust and torque by around 5% and 4% respectively.

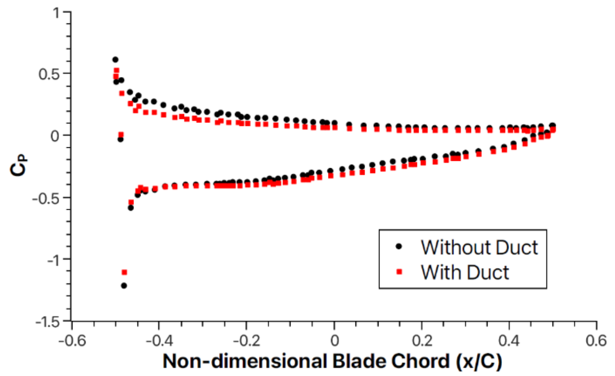

Fig. 10: Effect of the partial duct on pressure coefficient over the blade section at 0.7R in the model scale at J=0.2.

The variation of blade thrust and torque over a single revolution is studied using unsteady simulations. Fig. 11 shows the thrust coefficient of a single propeller at J=0.5 as a function of its angular position. It can be hypothesized that the effect of the flow acceleration due to the partial duct is maximum when the blade is closest to the duct (0-degree position), resulting in the minimum value of thrust coefficient. The thrust coefficient gradually increases with the change of angle as the distance of the blade increases from the elliptical duct. The maximum thrust is observed in the range in which the blade operates in the non-ducted condition.

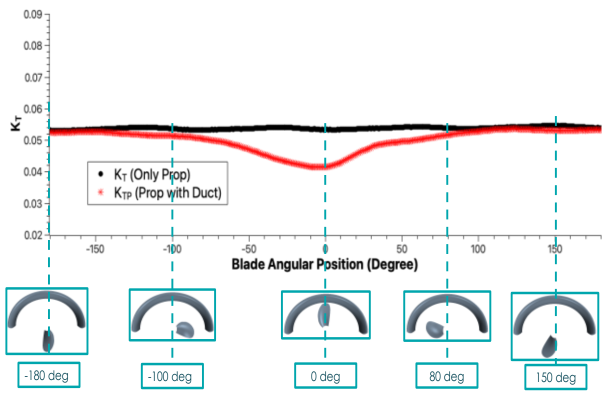

Fig. 11: Variation of the thrust coefficient of a single propeller blade over one revolution.

A critical aspect in the hydrodynamic performance estimation of ship propellers and associated energy saving devices is the effect of Reynolds number, often mentioned simply as the scale effect. Especially for ESD, the change in local flow and boundary layer between the model and full scales may result in differences in the corresponding percentage gains. This aspect has been highlighted in a benchmark study on the scale effects of a pre-swirl duct on KVLCC2 (Anderssen et al. 2022) where propeller modeling and wake prediction were identified as critical aspects in the prediction process.

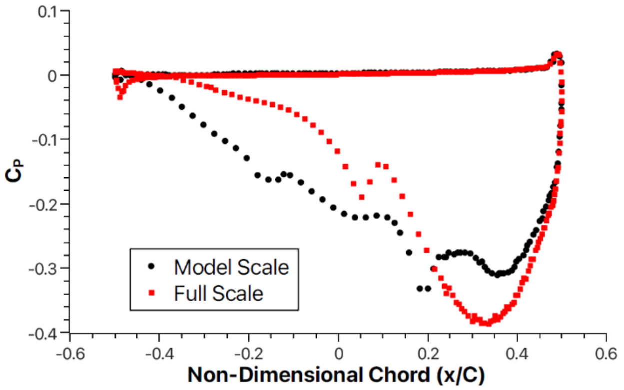

Fig. 12: Pressure coefficient over the partial duct section at 45° in model and full scales at J=0.2.

For ducted propellers, scale effects studies conducted by Abdel-Maksoud and Heinke (2002) and Bhattacharyya et al. (2016) have shown that, in general, the duct thrust increases and the propeller torque decreases in the ship scale as compared to model scale values. Considering the propeller and partial duct configuration used in the present study, similar observations have been made for the duct thrust. CFD simulations in open water conditions were conducted for the propeller and duct dimensions scaled 20 times. The duct thrust increased by more than 10% in the full scale resulting in a 3% higher open water efficiency. This can be explained using the comparison of pressure coefficient plots over the duct section in the model and full scales (Fig. 12). However, it is also observed that the scale effects on propeller thrust and torque are not significant for the selected propeller design.

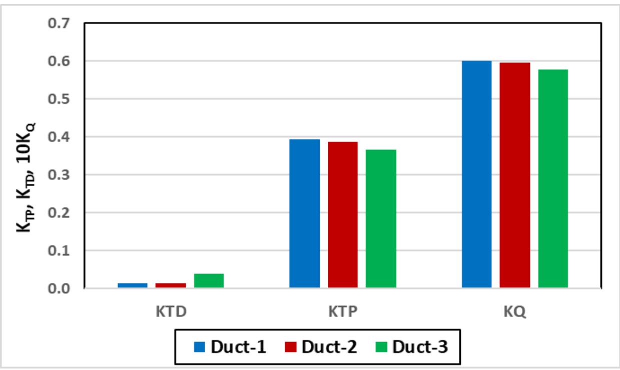

Fig. 13: Duct thrust coefficient, propeller thrust and torque coefficients for the three duct designs in open water condition at J=0.2.

The design of the partial duct is based on the primary intended function, which is to provide flow acceleration on the propeller plane in the region above the propeller axis where the wake fraction is higher. The basic purpose is in line with that of a pre-swirl duct having a diameter smaller than that of the propeller and providing flow acceleration in the upper half of the propeller disc. However, as the partial duct is located over the propeller, the hydrodynamic interaction between them is stronger, and it is important to estimate its influence on the open water performance.

Considering the novelty of the design, it is also imperative to study how the duct configuration impacts the open water performance.

For this purpose, three duct designs with varying major/ minor axis diameter ratio- 1.5 (Duct-1: original design), 1.25 (Duct-2), and 1 (Duct-3: semi-circular design) are considered. A comparison of the open water performance is presented in Fig. 13. It is observed that the fully semicircular duct (Duct-3) reduces the propeller thrust and torque in comparison to the other designs.

The pressure coefficient plots over the three ducts (Fig. 14) show that the leading-edge suction peak is much stronger for the semi-circular design (Duct-3). For the selected vertical section at 0 deg, the Duct-1 having the highest axis diameter ratio of 1.5 generates the maximum negative suction aft of the propeller near the duct trailing edge.

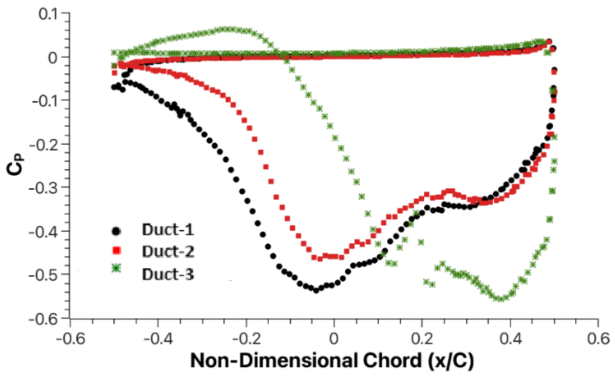

Fig. 14: Pressure coefficient over the section at 0° for the three partial duct designs in model scale at J=0.2.

The vorticity fields around the propeller-duct system for the three duct designs are shown in Fig. 15. The high vorticity regions are observed around the propeller blade tips and the edges of the duct. The vorticity magnitude is higher for the ducts with lower diameter ratio where the distance from the blade tip to the duct inner surface is lower at the edges. Higher periodic vortex shedding should be linked with greater pressure fluctuations over the duct, and hence the geometric parameters of the duct design like section profile, aspect ratio, and diameter ratio need to be suitably optimized.

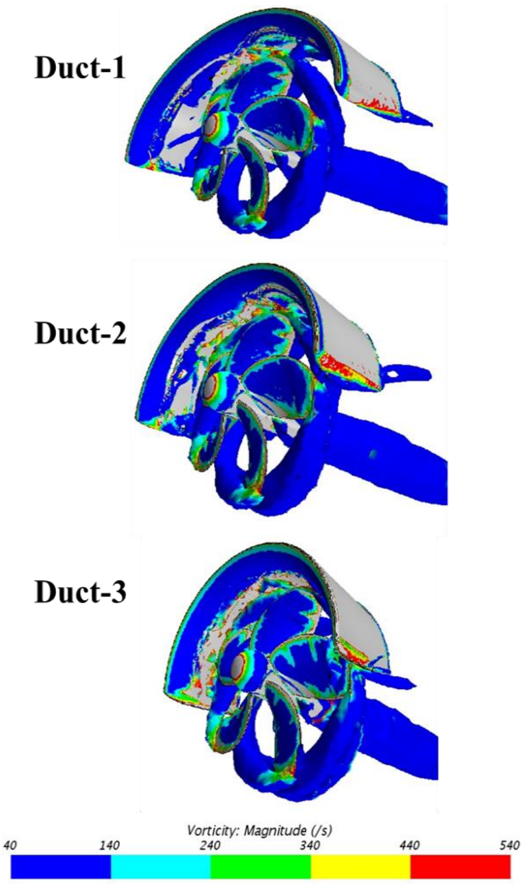

Fig. 15: Vorticity fields around the three duct designs at J=0.2 in the model scale.

In order to have an initial estimate of the performance of the partial duct in behind-hull condition, model propulsion tests are performed for a twin-screw inland vessel fitted alternately with and without the duct around the propellers (Fig. 16). It had been found that the partial duct resulted in the increase of the propeller thrust/ torque ratio by around 3% at a similar rpm in the self-propeller condition. Here, the aim has been to establish a simplistic comparison between the ducted and non-ducted cases without aiming for a detailed estimation of propulsive coefficients and vessel powering.

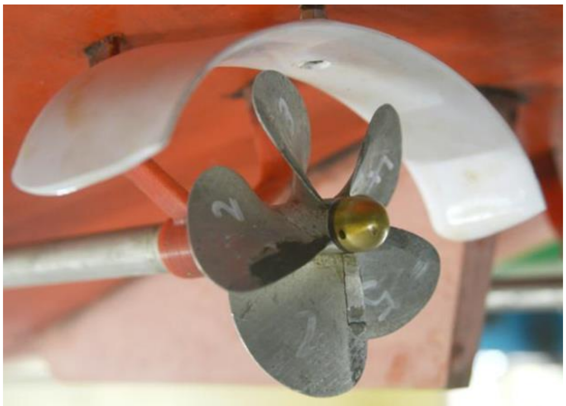

Fig. 16: The partial duct fitted over the propeller during the model self-propulsion tests.

It must be noted here that the present study provides some preliminary estimates of the hydrodynamic impacts of a partial accelerating duct on the propeller performance. On the other hand, aspects like stern design, support structure configuration and hull-propeller clearances will come into play for the implementation of the partial duct for practical applications. Also, propellers used in ducts generally have wide blade tips (Kaplan type), and hence it would be of interest to look into other propeller designs to understand the effectiveness in the partially ducted operation.

Considering the specific partial design, it will also be of interest to study the impact of the duct on the nominal wake distribution on the propeller. Though it is considered a part of the propulsion system, such an investigation should help in tailoring the duct design for improved wake characteristics and powering performance.

5 CONCLUDING REMARKS

The influence of a partial accelerating duct of semi-elliptic geometry on the open water performance of a 5-bladed propeller is investigated using CFD methods. Three duct designs based on different major axis/ minor axis diameter ratio are investigated. The thrust fraction from the partial duct is found to be much lower than that of a ducted propeller of similar section and aspect ratio. The propeller thrust and torque in open water are reduced due to the flow acceleration caused by the duct. Depending on the duct design, the resulting open water efficiency is 0.5-3% higher for the B5-75 propeller at high propeller loadings and decreases drastically with increasing advance coefficient as compared to the non-ducted condition. The Reynolds scale effects are more prominent on the duct thrust as compared to propeller thrust and torque.

Considering future research work, a comprehensive study including model tests and full-scale self-propulsion simulations needs to be performed to provide a realistic estimate of the partial duct on vessel powering.

ACKNOWLEDGEMENT

The authors would like to acknowledge the funds received from Naval Research Board, Govt. of India (Project: NRB/4003/PG/421) for performing some parts of the investigations presented here.

REFERENCES

Abdel-Maksoud, M. and Heinke, H.J. (2002), "Scale Effects on Ducted Propellers," Proceedings of 24th Symposium on Naval Hydrodynamics, Fukuoka, Japan.

Andersson, J. et al. (2022). 'Ship-scale CFD benchmark study of a pre-swirl duct on KVLCC2', Applied Ocean Research Vol. 123, 103134.

Bhattacharyya, A. Krasilnikov, V. Steen, S. (2016) 'A CFD-based scaling approach for ducted propellers', Ocean Eng., 123, 116-130

Bhattacharyya, A. (2020). 'Partial accelerating duct around a marine propeller' India Patent No. 506503

ITTC 7.5-03-01-01 (2008). 'Uncertainty Analysis in CFD Verification and Validation Methodology and Procedures'

Korkut, E. (2006). 'A case study for the effect of a flow improvement device (a partial wake equalizing duct) on ship powering characteristics' Ocean Engineering, 33(2), 205-218.

Mewis, F. (2009). 'A Novel Power-Saving Device for Full-Form Vessels', First International Symposium on Marine Propulsors, SMP'09, Trondheim, Norway.

Sasaki, N., Aono, T. (1997). 'Energy Saving Device ''SILD'', Journal of Shipbuilding Vol. 45, No. 135, Japan.

Schneekluth, H. (1986). 'Wake Equalising Ducts', The Naval Architect, London, UK.

Terwisga, T. (2013) 'On the working principles of Energy Saving Devices'. Third International Symposium on Marine Propulsors, SMP'13, Launceston, Australia.