COUPLED RANSE AND LIFTING LINE THEORY FOR ESTIMATION OF SHIP PROPULSION FACTORS

(DOI No: 10.3940/rina.ijme.2018.a3.485)

K Ramesh, and **I S Makkar**, Naval Construction Wing, Department of Applied Mechanics, Indian Institute of Technology, Delhi, India

SUMMARY

Advances in Computational Fluid Dynamics (CFD) techniques through the development of the Reynolds-Averaged Navier-Stokes Equations (RANSE) have assisted in estimation of resistance and propulsion characteristics of ships to a reasonable level of accuracy. The aim of this paper is to test and demonstrate the capabilities of the coupled RANSE and Lifting Line theory for undertaking ship resistance, propeller open-water and self-propulsion simulations. Further, parametric studies for generation of numerical propeller design sheets and optimisation of propulsive efficiency using the coupled simulation approach has been discussed. Commercial CFD solver "M/s Flowtech - Shipflow" has been used for the study. Initially, some benchmark experimental/numerical model results are validated with the results of the CFD simulations and then, further parametric analyses have been undertaken with the KRISO Container Ship and the KP505 Propeller. The numerical propeller series and the preliminary study methodology for optimization of location of propeller disc behind the ship's hull are being proposed as an effective concept/feasibility design stage tool for estimation of ship propulsion characteristics.

NOMENCLATURE

υ	Kinematic viscosity (N s m^{-2})
ρ	Density of water (kg m^{-3})
P	Pressure (N m^{-2})
R_n	Reynolds' Number
F_n	Froude Number
w	Wake Fraction
t	Thrust Deduction Factor
η_R	Relative Rotative Efficiency
C_T	Total Resistance Coefficient
C_F	Frictional Resistance Coefficient
C_R	Residuary Resistance Coefficient
C_A	Roughness Allowance
C_{AA}	Air Resistance Coefficient
C_{APP}	Appendage Resistance Coefficient
R_T	Total resistance (kN)
ITTC	International Towing Tank Conference
QPC	Quasi-Propulsive Coefficient
OPC	Overall-Propulsive Coefficient

1. INTRODUCTION

1.1 CFD IN HYDRODYNAMICS

Aiding in the study of the complex interactions between the hull and the propeller, Computational Fluid Dynamics (CFD) analysis is now an important tool in shipbuilding assisting the designer asses the design and its performance through optimization of hull forms, and appendages. With its multitude of advantages like providing quick and economic optimization solutions at an early design stage, come certain flaws like the need for validation of the results obtained. Keeping this factor in mind, CFD and experiments have thus, in parallel, started complementing each other in the prediction of ship hydrodynamics (Bhattacharya & Sha, 2010). Currently, model testing remains the mainstay in the performance prediction of ship-propeller systems prior to finalisation of the hull form but these reliable methods come with their own set of limitations. Ship CFD has fallen behind other industrial fields due to the inherent complexity in the geometry of ships as well as the complexity of the conditions at sea to which the ship may be exposed to during its service life. However the utility of CFD in analysing and predicting the ship performance in such complex practical situations has become a remarkable breakthrough with technological developments.

1.2 THE SCOPE OF WORK

The prediction of engine power required and the selection of the engine are vital components in ship design and any lapse in this regard can result in huge penalties during the service life of the ship. This in a broad sense encompasses, the efficiency of the propulsion train starting from the engine to the propeller, the efficiency of the propeller on its own and also the efficiency of the entire ship-propeller system as a whole. The present work aims at the use of CFD RANSE Solver "Shipflow" to study the flow around the ship's hull and obtain resistance and self-propulsion characteristics of benchmark models and extending the study through a parametric analysis to generate numerical propeller series charts and also to optimize the location of the propeller purely from a self propulsion point of view. The study thus emphasizes the ability of numerical methods to be used as initial design tools in self propulsion optimization methodologies.

2. LITERATURE SURVEY

2.1 PREVIOUS WORK ON THE SUBJECT

- Brizzolara, Villa and Gaggero (2008) conducted an extensive validation study about the results obtained using two CFD methods for marine propeller design at the Marine CFD group of the University of Genova, Italy. A series of results were obtained for 5 standard test propellers, simulated for a uniform onset flow over a range of advance coefficients using both panel method and RANSE and then compared with experimental values, with the RANSE solver demonstrating a slightly better accuracy.

- Choi, Min, Kim, Lee and Seo (2010) conducted resistance and propulsion simulations of different types of merchant vessels. ITTC78 method was used for extrapolation and the differences in the results with the experimental values were analysed to reveal similar tendencies, thus suggesting the potential to apply the computational method in predicting performance at the hull design stage.
- In the Workshop on CFD in Ship Hydrodynamics (2015) held in Tokyo (Kim, 2015), resistance and self-propulsion characteristics of the KRISO Container Ship were discussed using results from various CFD codes in towing tanks all over the world and a comparative analysis was obtained with respect to experimental values.
- Boucetta and Imine (2016) used the RANSE solver in Fluent and a DTMB 4148 hull form to analyse effect of number of blades, skew and blade thickness on the propulsion characteristics of a conventional marine propeller. The number of blades influences the propeller efficiency with the maximum efficiency being obtained for a propeller with 4 blades over a parametric study of 3 to 5 blades. Also, it was observed that for the 3 bladed propeller, an increase in thickness has a positive effect on the propeller efficiency and a positive skew to the blade angle improves open water efficiency at higher values of advance coefficient.
- Makkar et al (2013, 2014) undertook detailed numerical self-propulsion simulation for known CFD benchmark model - KVLCC 2. The results of wake were found to be sensitive to the Froude number and grid arrangement selected for the study. The wake distribution was also reported to be sensitive to the propeller location. High Froude number vessels posed a challenge for the numerical tool.

2.2 METHODOLOGY

The methodology for self propulsion calculations is as per standard naval architecture practices in the world. The resistance of the ship is first calculated with the frictional component of resistance obtained using the ITTC 1957 friction line and then the ITTC 78 method is employed for full scale extrapolation. The open water characteristics of the propeller are obtained and this when combined with the resistance of the ship generates the self propulsion results for a given hull-propeller configuration. Once these results are validated, further studies are conducted to expand the scope of the study through parametric analyses. The propeller theory used is the lifting line theory. The lifting line method is a mathematical method to compute the lift generated in a wing. The method assumes the propeller blade sections to be replaced by a single line vortex that varies in strength across sections. The line, about which the vortices act, has radial continuity. The mean circumferential wake distribution takes into consideration the inflow to the propeller which has a significant effect on its characteristics. Lifting line theory supposedly offers moderate reliability for standard propeller geometry amongst the wide array of propeller theories available.

3. THE TOOL AND GEOMETRIES

3.1 INTRODUCTION TO SHIPFLOW

The tool used for this study is the Swedish company Flowtech International AB's commercial CFD tool – Shipflow 6.3. The tool is custom made for the analysis of flow around ships and provides many powerful capabilities in the form of optimization algorithms. The software has various modules for various types of flow computations and panelling [8]:-

- XPAN – potential flow solver, gives output in the form of wave resistance, wave pattern, wave profiles, potential streamlines and pressure contours
- XGRID – grid generator for viscous computations, allows concentration of points where required and also flow computations for unsymmetrical cases.
- XMESH – panel generator for XPAN, allows generation of off- body points for computations.
- XBOUND – thin boundary layer computations, based on the momentum integral equation, gives output in the form of boundary layer thickness, momentum thickness, displacement thickness etc.
- XCHAP – finite volume RANSE solver using turbulence models (k-ω SST), gives output in the form of velocity and pressure field, turbulent kinetic energy and resistance coefficients.

XCHAP is a RANS (Reynolds-Averaged Navier-Stokes) solver which uses k-ω SST and EASM as turbulence models. It is finite volume based. In Shipflow calculations, the k-ω turbulence model is employed in the stern where the viscous effects are predominant and there is significant flow separation in self-propulsion simulations. Second order accuracy is obtained by using a Roe scheme discretization for the convective terms and a second order explicit defect correction. The rest of the terms are central difference discretized. Once a local artificial time-step is added to the equations an ADI-solver is used to solve the discrete coupled equations. Solving the RANS equations results in the time averaged velocity and pressure. Since the time fluctuating velocity and pressure are in general much smaller in amplitude, knowing the average will usually suffices. A zonal approach is employed in Shipflow where the Navier – Stokes equations are solved by dividing the flow domain into 3 zones differing in the flow characteristics. The potential flow region towards the front of the ship is solved using the panel method in XPAN. The boundary layer region is solved using momentum integral equations using XBOUND. In the aft of the ship where the boundary layer is thick, viscous flow solver of XCHAP is used. If global approach is used, the equations are solved in the entire flow domain. In the present study, the Zonal approach of Shipflow has been incorporated.

3.2 MESHING AND THE CONTROL VOLUME BOUNDS

Shipflow offers both automatic and manual meshing options. The meshing by default is structured and coarse. Shipflow allows options for very coarse, coarse, medium and fine meshes with the number of cells increasing with fineness. The increase in the fineness comes with a compromise on the time taken for calculations as the simulations take much longer time with a finer mesh that with a course mesh but give a more precise result. There is an additional option to make the mesh denser in the aft part of the ship where the flow is predominantly turbulent. Shipflow doesn't use conventional rectangular faces as bounds for its control volume. It defines its front and aft faces are quadrants of a circle, the top and side faces are rectangles connected by a quarter cylindrical planes as shown in Figure. 2 By default, the inflow and outflow planes are located 0.5 L and 1L respectively ahead and behind the ship, where L is the length of the ship. The side and bottom extend to 3 times L.

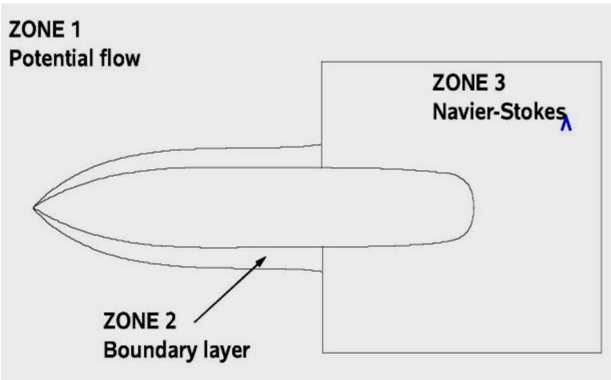

Figure 1 – Zonal approach in Shipflow (Bhattacharya & Sha, 2010)

Table 1- Hull form parameters

Model Used	Length (m)	Scale Ratio	Froude No	Full Scale Speed (kn)
KCS	230	31.6	0.26	24

3.3 GEOMETRIES USED

The geometries used for the studies are benchmark merchant ship model, the KRISO Container Ship (KCS) and the KP505 propeller primarily due to the availability of data from model testing tanks all over the globe for the validation of the CFD results. Particulars of the hull form and the propeller used are given in Table 2 and Table 3 respectively.

Validation studies are conducted with the KCS hull form and the KP 505 propeller as the validated results are available for these geometries from towing tanks all over the world. Further parametric analyses can be conducted on these validated values. The other hull forms can be used to analyse the application of this CFD tool for different hull and propeller geometries.

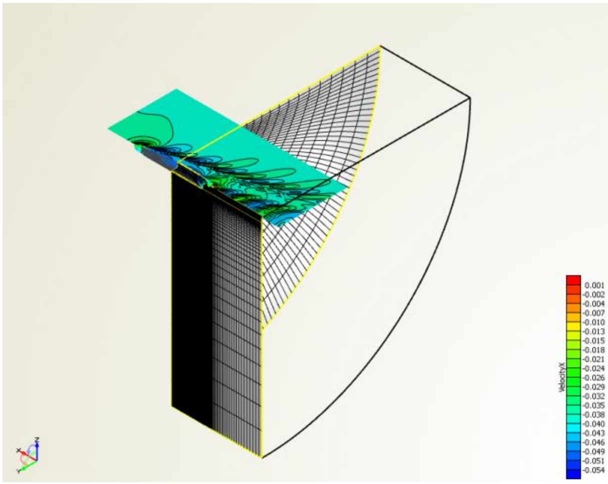

Figure 2 – Control volume in Shipflow

Table 2- Parameters of KCS Hull form

Parameter	KCS Full Scale	KCS Model Scale
L_{BP} (m)	230	7.28
B_{WL} (m)	32.2	1.02
D (m)	19	0.6
T (m)	10.8	0.34
Displacement (m^3)	52030	1.65
C_B	0.65	0.65
C_M	0.985	0.985
LCB (%) Fwd +	-1.48	-1.48

Table 3- Parameters of KP505 Propeller

Parameter	Full Scale Value	Model Value
Type	FP	FP
No. of Blades	5	5
D (m)	7.9	0.25
P/D	0.997	0.997
A_E/A_0	0.8	0.8
Hub Ratio	0.18	0.18

4. RESULTS AND ANALYSIS: NUMERICAL SIMULATIONS

4.1 VALIDATION STUDIES

The KRISO Container Ship is a benchmark model for resistance and self- propulsion tests through experimental and CFD studies in towing tanks all over the world. In the Tokyo Conference on CFD in Marine hydrodynamics (Kim, 2015), experimental fluid dynamics and CFD results using various RANSE solvers from towing tanks from across the globe were presented. These results were used to validate the resistance and self propulsion characteristics of the KCS hull, obtained using Shipflow as part of the current study, using 5 different grid sizes thus forming part of a Grid refinement study and hence obtaining the optimum grid size for further computations. The Grid no. 4 as given in Table 2 is chosen as the optimum grid for further simulations on this model.

The Grid number 4 is then used for model scale CFD analysis over the entire operating regime of the KCS hull and the results are as plotted in Figure 3. The mean percentage standard deviation of the optimized grid is 1.86 % compared to 1.88 % of the mean of CFD results from towing tanks. This result is well within the acceptable limits of uncertainty analysis as per standard CFD practices in ITTC standard norms.

Table 4 – Stages of Grid Refinement

S No.	Grid	No. of Cells	C_T	%Error from EFD (C_T= 0.0035)
1	Mean Of CFD results from Towing Tanks	-	0.00351	0.286
2	Shipflow Grid 1	220304	0.003498	0.057
3	Shipflow Grid 2	1007454	0.003483	0.485
4	Shipflow Grid 3	1770394	0.003495	0.143
5	Shipflow Grid 4	2700874	0.003499	0.029
6	Shipflow Grid 5	4719304	0.003497	0.086

The presence of the propeller behind the ship alters the flow around the hull and thus the wake fields obtained with and without the propeller behind the hull, called effective and nominal wake respectively, are expected to be different and are compared as given in Figure 4

The KP505 Propeller was used for open water simulations as part of another validation study. The results were obtained with model test open water data as shown in Figure 5. Shipflow gives good results within a range of J values from 0.1 to 0.9 with the mean standard percentage deviation from the experimental values being 1.73 % for K_T (Thrust Coefficient), 3.41% for 10 K_Q (Torque Coefficient) and 0.47% for η_0 (Open Water Efficiency). Once the Open Water Characteristics were validated, the KP 505 propeller was added to the KCS hull and Self Propulsion Simulations were run. The resulting thrust and torque in non-dimensional forms were compared with experimental and mean CFD results of towing tanks as shown in Figure 6. The various other results given by the self- propulsion simulation in Shipflow like wave profile, pressure contours and wake velocity component plots are as given in Figures 7-10. Figure 7 shows the coloured contour plots of non-dimensional (w.r.t Ship's length) wave elevations near the ship's hull. Figure 8 shows the coloured contour plot of non-dimensional pressure (pressure coefficient) on the ship's hull and the dark black lines indicate the streamlines along the hull. Figure 9 shows the tangential velocity component of the ship's velocity as a fraction of the total velocity over various locations on the propeller disk in the form of a graph rather than a wake plot. Figure 10 shows the transversal velocity components as a ratio of the total velocity (indicated by circumferential lines) at various angles over the propeller blade (radial lines) and the arrows point in the direction of the transversal velocity at that particular location on the propeller disk.

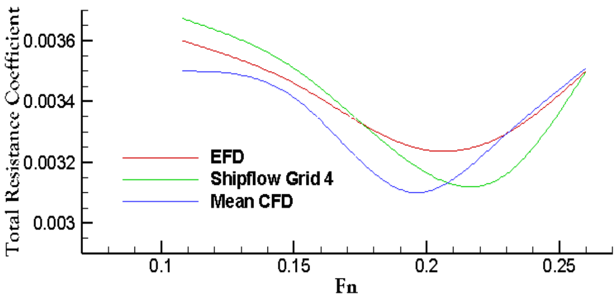

Figure 3 – Comparison of resistance results

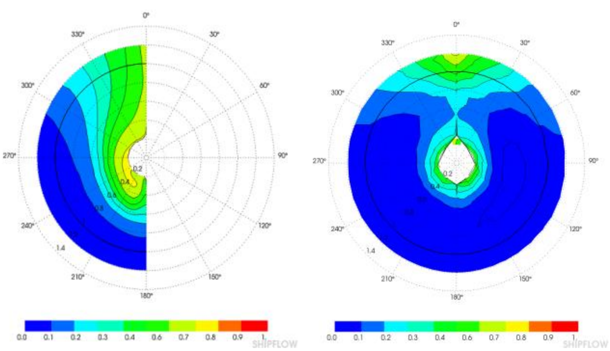

Figure 4 – Nominal and effective wake plots

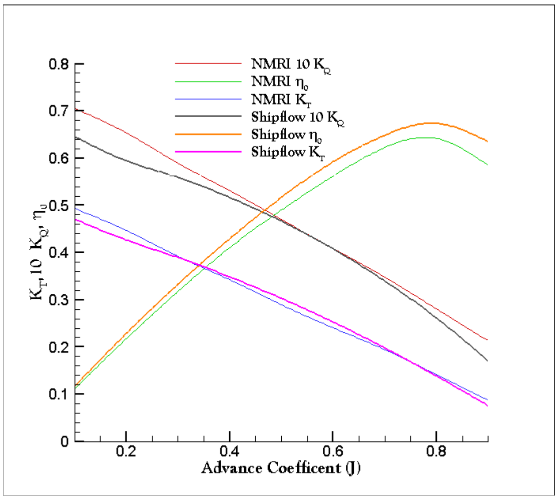

Figure 5 – Comparison of Open Water Characteristics of KP505 Propeller

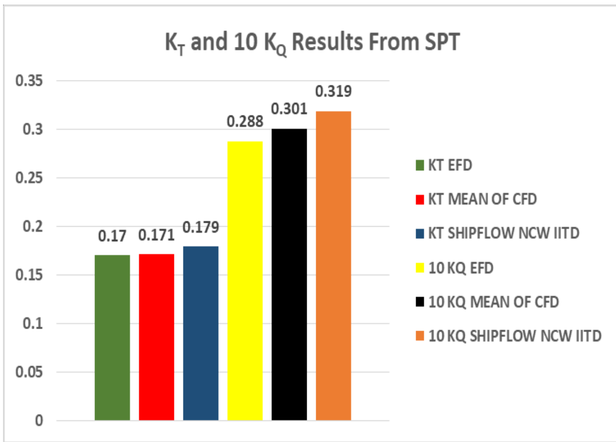

Figure 6 – Comparison of Self-Propulsion Parameters

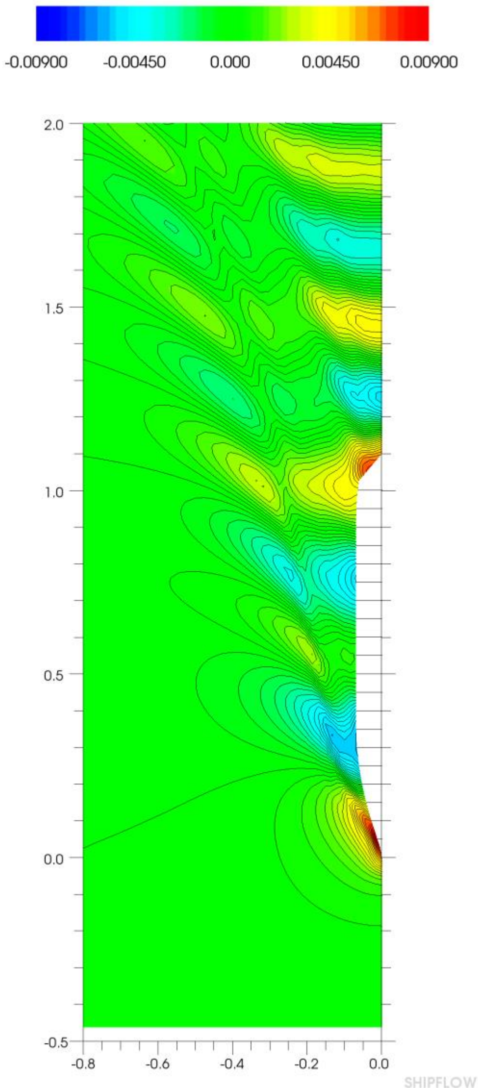

Figure 7 – Free Surface elevation contours

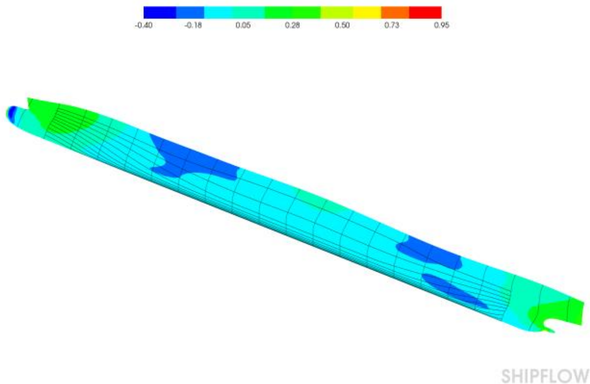

Figure 8 – Cp distribution and potential streamlines on the hull

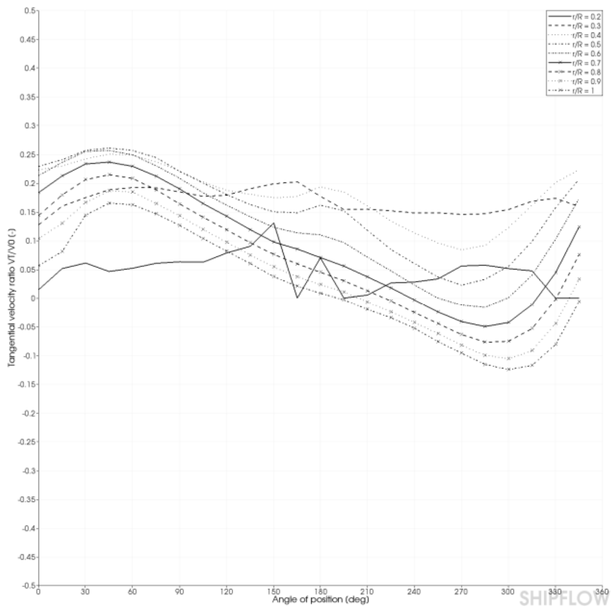

Figure 9 – Tangential velocity ratio plot over the propeller disk

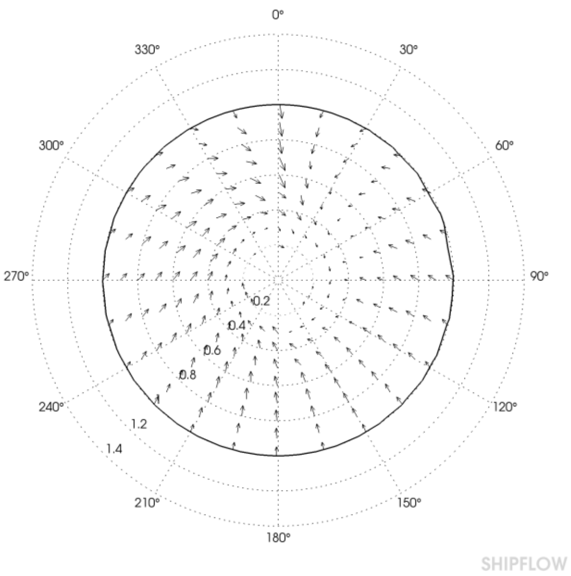

Figure 10 – Transversal velocities in propeller disk

4.2 PARAMETRIC ANALYSES

The validation studies conducted showed the capability of Shipflow to produce very close results to experimental values. Hence, the tool was used to conduct parametric analyses and a few further studies which can be developed as a powerful tool to assist concept design stages of ships. The number of blades in the KP 505 propeller was changed and open water charts were obtained for 4, 5 and 6 bladed propellers, keeping the diameter of the propeller fixed. Thus a numerical series chart was obtained which can be developed to provide an early stage concept design tool for propeller selection for ships. The chart generated for the KP 505 propeller is shown in Figure 11. Same parametric analysis by changing the number of blades was done to conduct self-propulsion simulations with the KCS hull and the resulting Torque, thrust and efficiencies are compared in Figure 12.

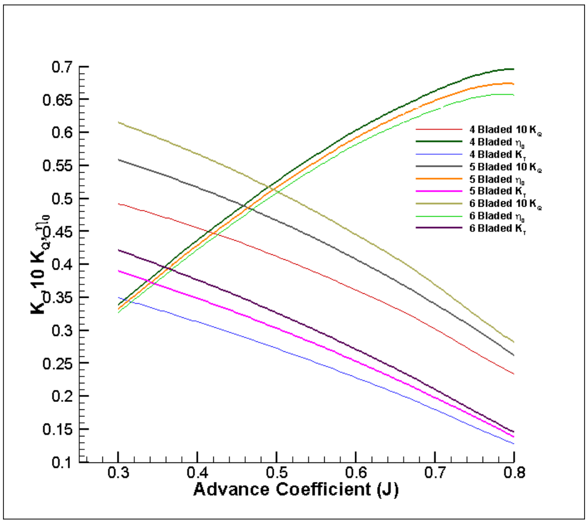

Figure 11 – Numerical Open Water Series

An optimization study was conducted to find out the optimum location of the propeller disk behind the aft perpendicular, keeping propulsive efficiency as the factor for comparison. With the distance of the centre of the disk kept at 4 m from the keel line, the propeller was moved ahead and behind its current location of 4.1 m, from 3.5m to 5m to obtain the maximum efficiency. The x location is optimized at 4.25 m. With x held constant at 4.1m the height of the propeller from the base line was varied from 2 to 5.5 m and the highest efficiency was obtained at 4.9 m as shown in Figure 13. The study is purely based on self-propulsion factors and how they are related to location of propeller disk, and other factors affecting optimum propeller location like stern hull form and vibrations etc. have not been studied.

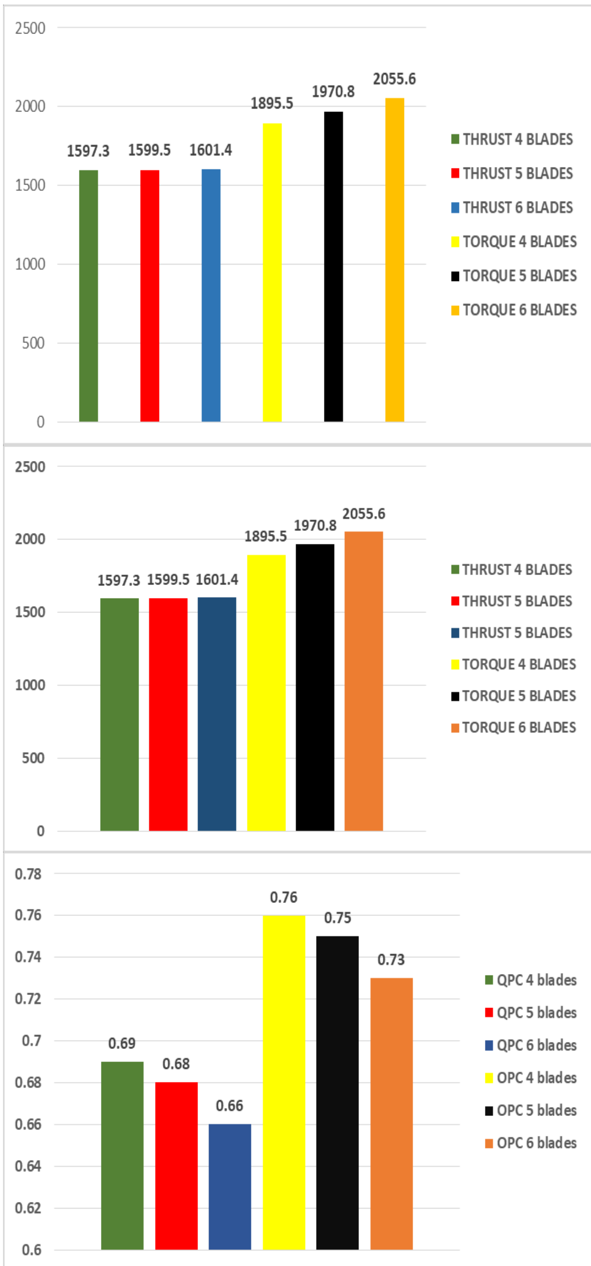

Figure 12 – Comparison of Self Propulsion Parameters for varying blade numbers (Thrust in kN and Torque in kNm)

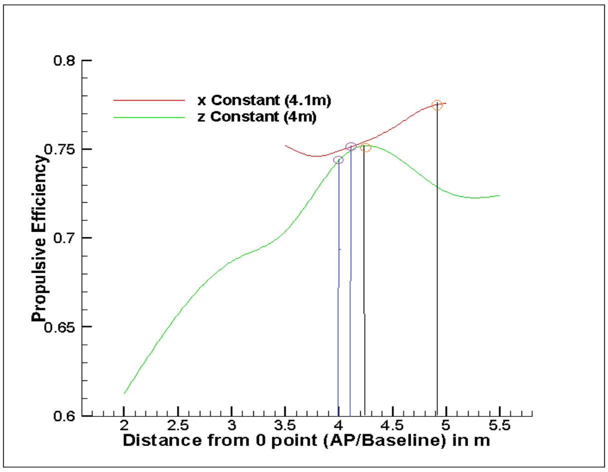

Figure 13 – Location optimization for propeller disc

5. CONCLUSION

The numerical self-propulsion optimization methodology was validated as the results were well within the acceptable limits as given in ITTC procedures. Parametric Analyses were conducted to obtain numerical open propeller series charts. Location optimization studies were conducted to obtain optimum location of propeller from a self-propulsive efficiency point of view. The possibility of using the tool to generate open water series charts gives a preliminary design stage tool to assist propeller selection.

6. ACKNOWLEDGEMENTS

This research was greatly supported by Mr. Lief Broberg and Flowtech Shipflow technical support team. We thank our colleagues from Indian Institute of Technology, Delhi who provided insight and expertise that greatly assisted the research.

We thank Cdr MP Mathew, Naval Construction Wing, IIT Delhi, for his assistance and for comments that greatly improved the manuscript.

7. REFERENCES

1. BHATTACHARYA, A; SHA, O.P., *Integration of CAD and CFD for Hydrodynamic Design of Ships*, INMARCO –INAvation, 2010.
2. BRIZZOLARA. S., VILLA. D., GAGGERO. S., *A systematic comparison between RANSE and Panel methods for Propeller Analysis*, Marine CFD group, University of Genova Department of Naval Architecture and Marine Engineering, Oct. 2008.
3. CHOI. J. E., MIN. K. S, KIM. J. H, LEE. S. B., SEO. H. W., *Resistance and Propulsion Characteristics of Various Commercial Ships based on CFD results*, Hyundai Heavy Industries Ltd., Republic of Korea, Elsevier Ocean Engineering Vol. 37,pp. 549-566, Feb. 2010.
4. KIM, J., *Report on the result of KCS Resistance and Self Propulsion*, Tokyo Workshop on CFD in Ship Hydrodynamics, Tokyo, 2015.
5. BOUCETTA. D., IMINE. O., *Numerical Simulation of flow around marine propellers*, Journal of Physical Sciences and Applications Vol. 6(3),pp. 55-61 2016.
6. MAKKAR, I.S., SHA, O.P.(2013), "*Numerical Self Propulsion Simulation and Propeller Induced Vibration Study*", Proceedings of International Conference on Ship & Offshore Technology: Technical Innovation in Shipbuilding, The Royal Institution of Naval Architects, IIT Kharagpur.
7. MAKKAR, I.S., VIJAYAKUMAR, R., "*Numerical Resistance & Self Propulsion Simulations for Early-Stage Ship Design Evaluation*", IWCEM DIAT, Pune, 2014
8. Flowtech International AB, *Shipflow 6.3 User Manual*.