INTERNATIONAL JOURNAL OF INNOVATIVE RESEARCH & DEVELOPMENT

ISSN: 2278 – 0211 (Online)

Numerical Investigation Of Stern Flow For Improved Hull Design

R D B Prasada Naidu Dabbi
Assistant Surveyor, Research & Rule Development division of Indian Register of Shipping, Mumbai, India
O P Sha
Professor and Head of the department (Department of Ocean Engineering and Naval Architecture), Indian Institute of Technology, Kharagpur, India

Abstract:

The International Maritime Organisation has already made Energy Efficiency Design Index (EEDI) as a design norm which requires looking at design from energy consumption point of view. The study of the flow in and around the stern region of a ship needs greater attention. This is because of the fact that, the flow in the stern region defines the performance of the propeller. A highly uneven flow at the stern may lead to cavitations, vibrations and decrease in the propulsive efficiency of the vehicle. Recent advances in computational fluid dynamics techniques and development of Reynolds-averaged Navier-Stokes equations based methods have enabled a reasonably accurate estimation of resistance and facilitated the design evaluation of aft-body form and appendages that are highly influenced by viscous effects. By integrating CAD and CFD techniques along with databases of model tests, it is possible to explore and develop efficient hull forms within specified time constraints.

1.Introduction

The current situations in the global economy has triggered and expanded the shipbuilding and shipping industry. There is high requirement for design and building both conventional and new types of ships with high performance. Simultaneously, environmental concern and a constant increase in the price of fuels has put more pressure on the designers to minimize the energy consumption, maximize the protection of the marine environment and maximize the efficiency of marine operations. The ship propulsion system has to be improved continuously to satisfy these requirements.In the recent past, the study of the flow in and around the stern region of a ship has gained much importance. This is because, the flow in the stern region defines the performance of the propeller. The quality of flow received by the propeller translates itself into propeller thrust producing efficiency and the level of vibration and associated noise. A highly uneven flow at the stern may lead to the propeller blades encountering rapidly varying axial flow velocities during rotation. This means a rapidly varying angle of attack and pulsating loads, which gives rise to vibration. In the present work, an attempt has been made to numerically simulate and analyse the interaction between hull and propeller using SHIPFLOW a RANS based solver. Numerical simulation and analysis of such interaction is initially carried out for a twin screw offshore patrol vessel where model test data for resistance (with and without appendages) and wake were available so as to validate the numerical model.

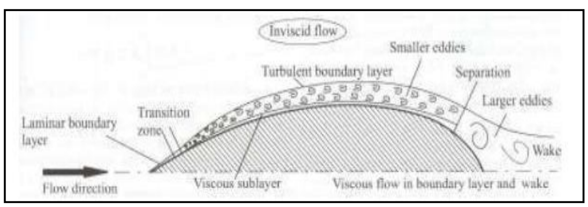

Figure 1a: Regimes of flow around a ship shaped body

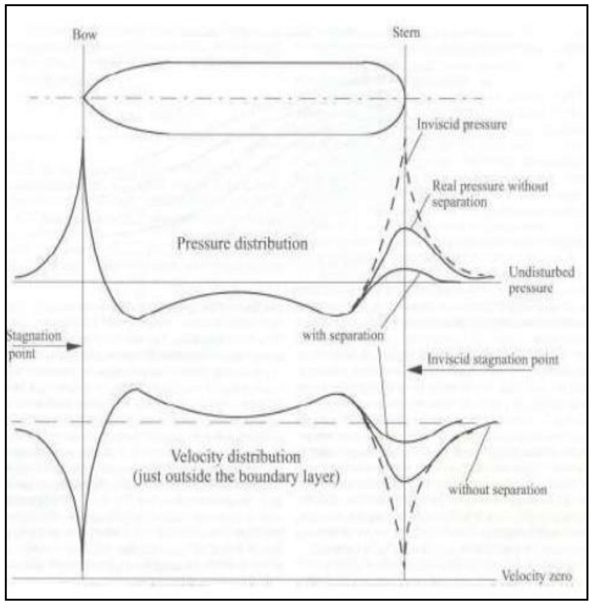

Figure 1b: Effect of fluid viscosity & flow separation on the flow at the stern of a ship

2.Software And Tools Used

MAXSURF (Design module): Design of hull form and its modifications.

MAXSURF (Hydrolink module): Generation of offset file, necessary for subsequent hydrodynamic analysis.

SHIPFLOWR: Hydrodynamic analysis.

AUTOCAD: Linesplan generation for model construction on a predetermined scale.

RHINO: For hull and propeller modeling

3.Literature Survey

In the last few years a great deal of research regarding hydrodynamic design and optimization of marine vessels has been done. The prime objective is the development of a hull form with minimum resistance and with a flow around it, which is least disturbed, both of which translate into least power requirement within the design constraints. In view of the present work, much literature survey was done for a favorable flow at the stern. By favorable, it means that the flow is as uniform as possible with the minimum axial and radial wake components.

Krüger et al. [1] in their paper have emphasized on the development of an efficient simulation process for an analysis of the hydrodynamic performance of vessels using RANS. Emphasis is given to the assessment of wake flows. Attention is given to robustness and process-efficiency aspects with respect to flow- physics and geometry modeling.

In order to reduce the effort of a RANS analysis, a process chain involving a ship-design system, semi-automatic grid generation as well as a RANS code was jointly developed between the institutes of Ship Design and Ship Safety (SSI) and Fluid Dynamics and Ship Theory (FDS) at the Hamburg University of Technology (TUHH).

Firstly, the bare-hull performance is analyzed with a potential flow method, which is substantially faster than a RANS computation. Several design variants were tested with respect to the wave resistance, which provided a first glance at the flow field around the ship and the expected pressure distribution on the hull. During the next step, the optimized hull form was assessed with RANS methods. Again, the analysis provided not only the resistance and the nominal wake field but also further

details of the flow field which were used for the optimization of the shape and the configuration of appendages.

Larsson and Raven [2], in their release named "Ship Resistance and Flow", introduced a numerical method for predicting the flow and resistance components of ships and marine structures. The theory is based on an approach where the flow is divided into three zones: the potential flow with a free surface, the boundary layer and the Navier-Stokes flow. A separate method is used for each zone, thus enabling a high resolution without excessive computer efforts. The usefulness and the accuracy of the system are demonstrated through a number of computed examples, for which comparison is made with measurements. Predicted quantities include resistance components, wakes and other local flow quantities, also in the presence of a working propeller.

It serves as a guide in the choice of hull shape and dimensions. The possibility on local improvement on the hull shapes has been impressed upon. Results of wake distribution for various aftbody shapes have also been presented. This paper gives a insight into optimization of stern from a hull efficiency and vibration point of view.

In the work done by VAN et al. [3], flow characteristics around practical hull forms have been studied. A comparison is made between Computational method (economic in computational cost and time) and experimental method (reliable results). These two approaches have been used to give a comparison between the measured and calculated values of resistance, wave elevation, limiting streamlines, velocity distribution, pressure distribution and numerical wake data, Carlton, J. S [5], through a handout named "Marine Propellers and Propulsion", has given a very deep insight on topics like "The propeller Environment", "The Wake field", "Ship Resistance and Propulsion", "Propeller blade vibration and noise" and "Wake nullifying devices". The knowledge of these topics was very much necessary for the course of this study. A systematic insight into each of these topics has provided immense contribution to the present work. This book very elegantly reveals the underlying principles of devices like Wake equalizing ducts, Grotheus Spoilers, Asymmetric Stern, etc. Sharma and Sha [6] presented the design method that combines and extends two famous theories i.e. Kracht(1978a) and Yim (1980) for a particular set of requirements within a narrow range of parameters. The method uses a reanalysis of an approximate linear theory with sheltering effect for resistance estimation, and recorrelation with statistical analysis via a non linear multivariate regression analysis from existing literature and tank test results available in the public domain. The optimization of design parameters has been done for the design speed. The effect of change in the speed has been discussed and suitably incorporated in the design process. In the present work, the effect of production constraints on the design of bulb has also been briefly examined. The results of this study are presented in the form of design parameters related to main hull parameters for a set of input data in a narrow range. The first six parameters have been derived by recorrelation with statistical analysis and the seventh parameter by reanalysis of an approximate linear theory with sheltering effect for resistance estimation. Finally, a design example, which includes tank test results of an additive bulbous bow for container ship has been presented.

Yusuke Tahara et al. [7] performed steady flow simulations for the Korean Research Institute for Ships and Ocean Engineering (KRISO) container ship (KCS) for towing and self- propulsion. The main focus in the present article is on the evaluation of computational fluid dynamics (CFD) as a tool for hull form design along with application of state-of-the-art technology in the flow simulations. Two Reynolds-averaged Navier-Stokes (RANS) equation solvers were employed, namely CFDShip-Iowa version 4 and Flowpack version 2004e, for the towing and selfpropulsion cases, respectively. The new features of CFDShip-Iowa version 4 include a single-phase level-set method to model the free surface and an overset gridding capability to increase resolution in the flow and wave fields. The new features of Flowpack version 2004e are related to a self-propulsion scheme in which the RANS solver is coupled with a propeller performance program based on the infinitely bladed propeller theory.

4.Shipflow(R)

SHIPFLOW (R) is a special purpose computer code for predicting the flow around ship and around its component, developed at Chalmers University of Technology and FLOWTECH International, AB. To solve the flow around the hull two different approaches, i.e. global and zonal approaches are available in SHIPFLOW(R). A global approach means that the Navier-Stokes equations are solved in the whole flow domain. A zonal approach means that the flow domain is divided into different zones based on the flow characteristics inside, as shown in fig1. In Zone 1, which is characterised as inviscid and irrotational, a potential flow method is used. This method computes wave resistance and induced resistance, and provides input to a boundary layer method in Zone 2, where transition and boundary layer parameters on the forward half of the ship are predicted. In Zone 3 where the boundary layer gets very thick, a RANS code is used.

SHIPFLOW(R) consists of five modules as shown in the table. Two modules, XMESH and XGRID, are used for mesh and grid generation and the other three modules are for flow analysis. XPAN is applied for potential flow, XBOUND for the boundary layer flow and XCHAP and XVISC for Navier-Stokes region.

SHIPFLOW(R) can generate overlapping grid mainly for appendages such as rudder, bracket and shaft. If special shapes are needed which SHIPFLOW can't generate, imported overlapping grids are allowed. The imported grids can be scaled, rotated, and translated to give them the desired size and position. The boundary of the fluid domain is represented by faces of the component grids where boundary conditions are specified. The boundary conditions available are no slip, slip, in-flow, out-flow and in-out conditions.

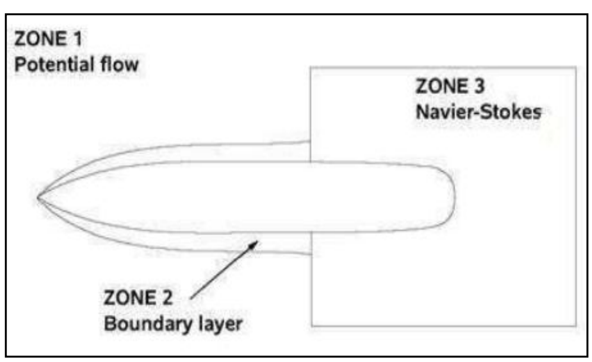

Figure 2a: Different zonal approaches in SHIPFLOW®

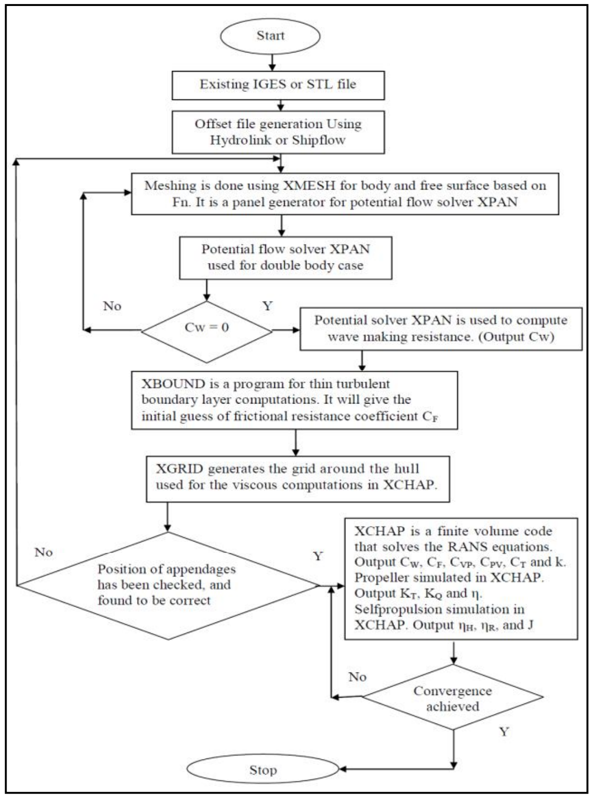

Figure 2b: Flow chart for Shipflow computations

5.Existing Vessel Design

The main particulars for the existing vessel are presented in the table below. These have been obtained from maxsurf:

Measurement	Value	Units
Displacement	1209.5	tonnes
Volume of Displacement	1180	m3
Draft to Baseline	3.197	m
Immersed depth	3.197	m
Lwl	69.217	m
Beam wl	10.719	m
WSA	745	m2
Max cross sectional area	27.684	m2
Waterplane area	521.83	m2
Cp	0.616	
Cb	0.497	
Cm	0.808	
Cwp	0.703	
LCB from zero pt. (+ve fwd)	34.058	m
LCF from zero pt. (+ve fwd)	31.264	m
LCB from zero pt. (+ve fwd)	49.255	%
LCF from zero pt. (+ve fwd)	45.21	%
Scale Ratio	11.59 /	

Propeller Data		
Measurement	**Value**	**Units**
Diameter of the propeller	2.75	m
Number of blades	4	
Propeller type	B-Series	
Pitch ratio	1.132	
Blade area ratio	0.974	
Boss diameter ratio	0.167	

Table 1: Main Particulars for the existing vessel

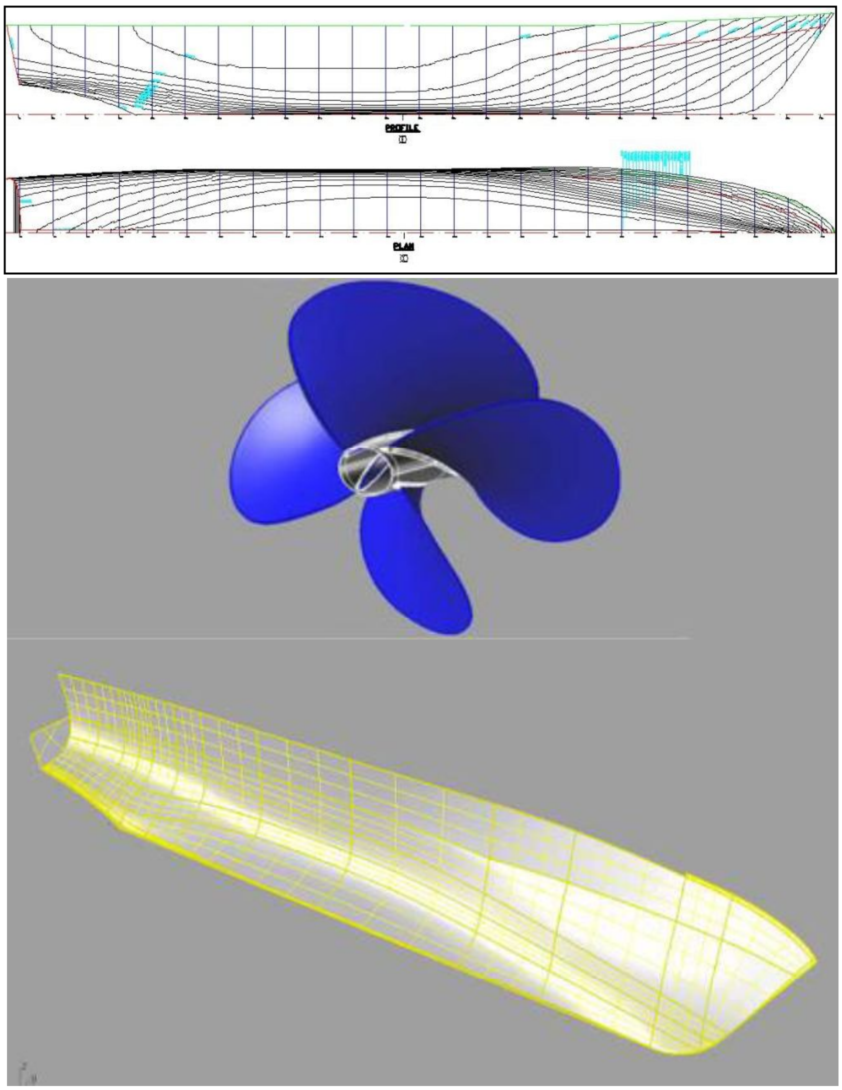

Figure 3: Lines plan and Hull of the offshore patrol vessel and Existing propeller of the vessel

6.Computations And Validations

The flow analysis of the initial design was done in CFD solver SHIPFLOW^R. The values obtained are wave resistance (from the potential flow solver- XPAN) and the viscous resistance (from the RANSE solver XCHAP). These computational results were then compared with the experimental results obtained from model test carried out at the towing tank, IIT Kharagpur and NSMB. The comparison was made on model. The whole procedure was followed for bare hull as well as appended hull resistance data comparison. The need for this exercise was to test the robustness of the code used. Similar methodology was followed for wake analysis. Wake results were used in propeller harmonics calculations.

Work can be broadly subdivided into following parts:

- Bare Hull resistance prediction for model at 3.197m draft and of scale ratio 11.59
- Bare Hull resistance prediction for model at 3.443 m draft and of scale ratio 21.62
- Appended Hull resistance prediction for model at 3.443 m draft and of scale ratio 21.62
- Wake at the propeller plane
- Propeller open water characteristics
- Self propulsion Simulation
- Propeller harmonics

6.1.Resistance Validation

The bare hull resistance values computed for model through CFD tool at a draft of 3.197 m and with scale ratio 11.59 and are compared with the tank test results obtained from NSMB. The values were found well matching.

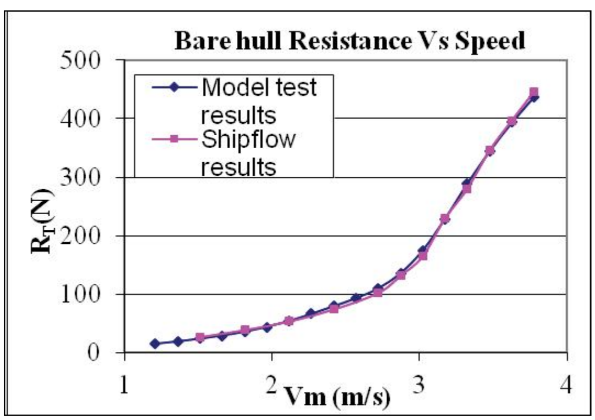

Figure 4a: Bare hull resistance characteristics for the OPV (at 3.197 m draft and of scale 11.59)

6.1.1.Appended Hull Resistance Validation

Overlapping grid technique is used to model the appendages (A-bracket, shaft and rudder) in Shipflow^{R}.

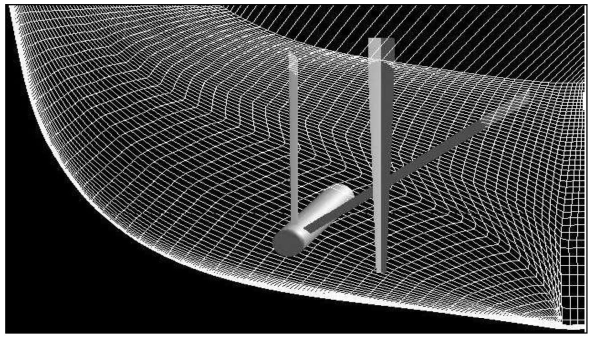

Figure 4b: Hull with appendages view from stern (Modeled in Shipflow)

The bare hull and appended hull resistance values computed by using CFD tool at a draft of 3.443 m and with scale ratio 21.618 are matching with the values obtained from the experiments conducted at the towing tank of IIT Kharagpur.

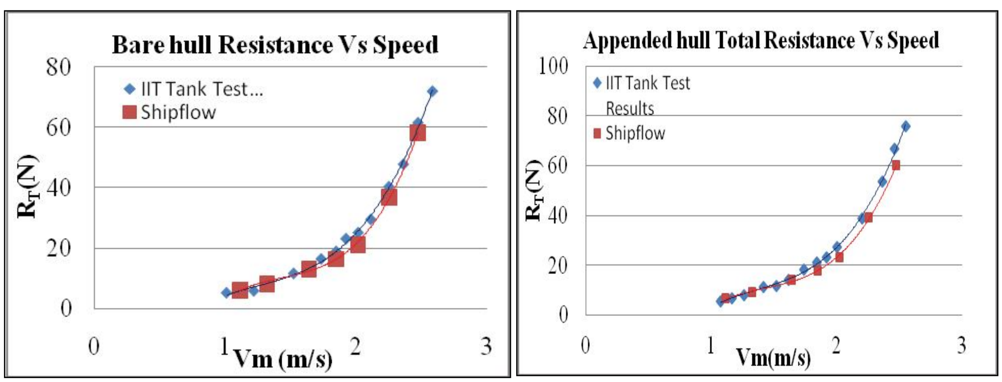

Figure 4c: Bare & Appended hull resistance characteristics for the OPV (at 3.443 m draft and of scale 21.618)

6.2.Wake Validation

The nominal wake at the propeller plane is then computed using Shipflow^{R}. The wake values are obtained from Shipflow^{R} at different r/R, compared with the wake values obtained from NSMB tank test results and found they all are in good agreement.

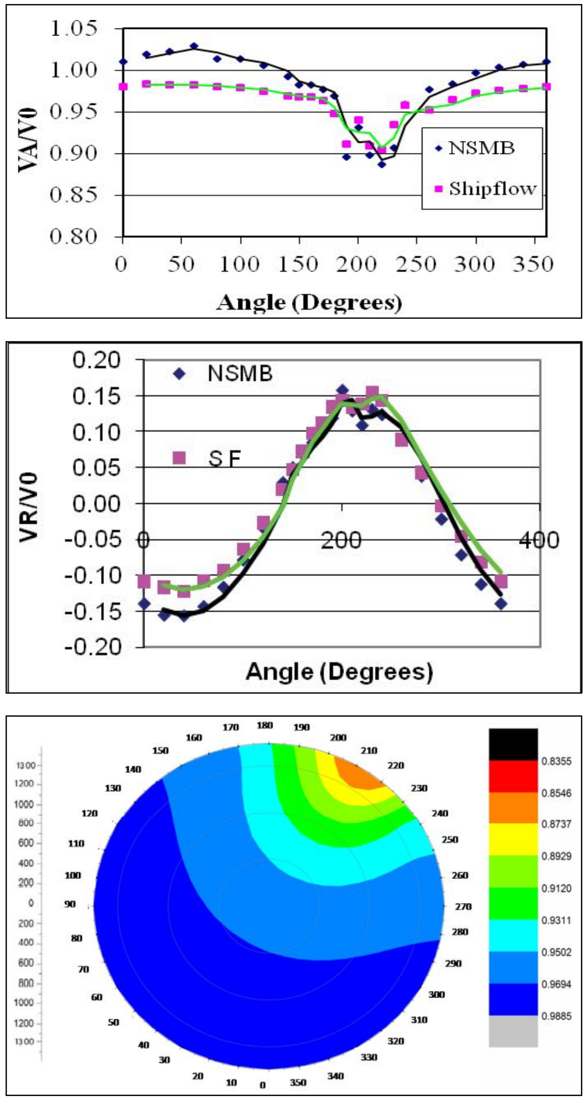

Figure 5: Axial wake at r/R = 0.843, Radial wake at r/R = 0.927 & Axial wake plot at the propeller plane

In case of any ship, model resistance and propulsion experiments are generally conducted at the design load condition and in some cases they will conduct at the estimated ballast condition also. But, these experiments take place in the laboratories, and in the due course we found that the ship is not being operated in exactly the same conditions of loading for which we have the model test results. Total power required to move a ship at some speed is estimated by total resistance coefficient, wetted surface area and the propulsive efficiency. The wetted surface area can be obtained at any draft from the hydrostatic curves. Where as, resistance and propulsion factors (wake, thrust deduction fraction and total resistance coefficient) at fractions of the designed load draft are different from the values measured on a model at designed draft.

So, subsequently, simulated runs in SHIPFLOW^R to obtain resistance and self propulsion results at fractions of designed draft. And thus the relation between the draft ratio $(T)_R$ (fractional draft) and wake is obtained. From these results we can approximate wake at any fraction of designed draft.

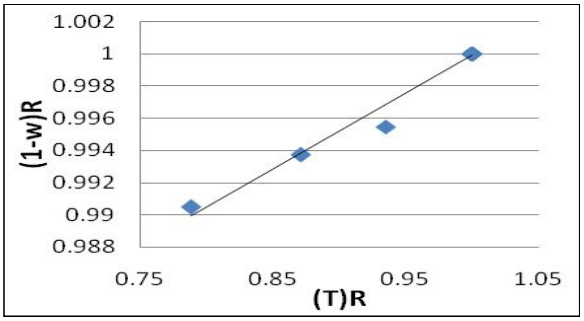

Figure 6: Variation of $(1\text{-}w)_R$ With $(T)_R$

Wake Fraction Ratio $(1\text{-}w)_R$:

$$(1\text{-}w)_R = 0.0475*(T)_R + 0.9525$$

$$\text{Where (T)R is Draft Ratio} = \frac{Draft}{Designed\ load\ draft}$$

6.3.Propeller Openwater

A propeller is simulated in Shipflow^R using lifting line method. Thrust coefficient, torque coefficient and open water efficiency (Openwater characteristics) at different advance coefficients obtained using CFD tool are compared with the values obtained from B-Series polynomials and found they all are well matching. % difference between openwater efficiencies varies from -2% to 2%, where as it is -8% to 6% for thrust and torque coefficients.

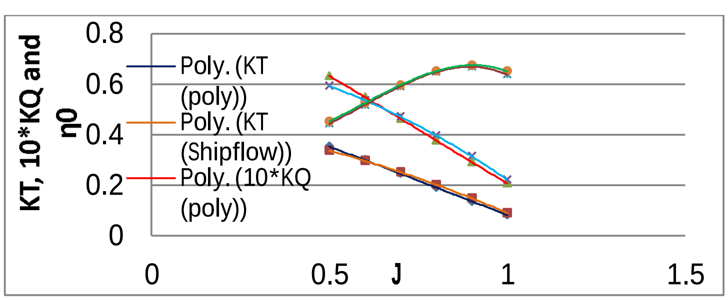

Figure7: Propeller Open water characteristics

6.4.Propeller Harmonics

Propeller harmonic calculations have been done and thrust and torque are computed at different radii for a blade taking openwater characteristics of the propeller at different angles. Later torque and thrust have been computed for remaining 3 blades as the considered propeller has 4 blades. All the individual values are summed up to get total torque and thrust at different angles. Graphs are plotted between thrust, torque and Angle.

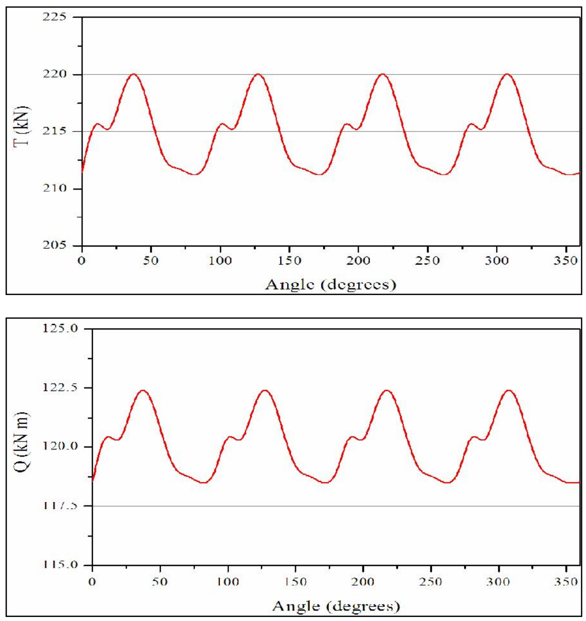

Figure 8: Thrust & Torque Vs θ at r/R = 0.843

Intermediate fluctuations are coming with a phase of 90^0, because of the presence of brackets and hull. If the thrust variation is high along radial and angular directions, propeller design is to be modified in design stage or to adopt design modifications to make flow uniform.

7.Conclusion

The work has yielded a validated numerical model for bare hull resistance, appended hull resistance, propeller open water characteristics, propulsive factors and stern flow. The numerical model applied in current work can predict the flow field and it is a good visualization tool for complicated flows e.g. in the stern region. It is very helpful for designers to understand and optimize complex flow problems and by integrating CAD and CFD techniques along with databases of model tests, it is possible to explore and develop efficient hull forms within specified time constraints.

8.Acknowledgement

The authors gratefully acknowledge the support of all the faculty members of the "Department of Ocean engineering & Naval Architecture" through their technical inputs and arrangement of necessary facilities as and when required. They also sincerely thank Mr. Leif Broberg and Mr. Magnus Ostberg of Flowtech International for their constant help and assistance with Shipflow^{R}

- **Nomenclature**

$(1\text{-}w)_R$ = Wake fraction ratio

η_H= Hull efficiency

η_o = Openwater efficiency

η_R= Relative rotative efficiency

C_F= Frictional resistance coefficient

C_T= Total resistance coefficient

C_W= Wave making resistance coefficient

J= Advance coefficient

K_T = Thrust coefficient

K_Q= Torque coefficient

R_F= Frictional resistance

R_W= Wave making resistance

R_T= Total resistance

T_R= Draft ratio

V_A= Advance velocity

V_R= Radial Velocity

V_0= Ship speed

V_m= Model speed (m/s)

9.Reference

1. Stefan Kruger, Manuel Manzke, Thomas Rung, Hendrik Vorholter, "Introduction of RANS-CFD into the Initial Design Process"
2. Larsson, L. and Raven H. C., "Ship Resistance and Flow", The Society of Naval Architects and Marine Engineers", 2010, USA
3. VAN, S. H.; KIM, W. J.; YIM, G. T.; KIM, D. H.; LEE, C. J., (1998b), Experimental investigation of the flow characteristics around practical hull forms., Proceedings, 3rd Osaka Colloquium on Advanced CFD Applications to Ship Flow and Hull Form Design, Osaka, Japan.
4. Valkhof.H and Hoekstra.M (1998), "Model tests and CFD in Hull form Optimization", SNAME Transactions, Vol. 106, 1998, pp. 391-412.
5. Carlton, J. S., "Marine Propellers and Propulsion", 2nd edition, Elsevier, 2007.
6. R.Sharma and O.P.Sha (2005) "Practical Hydrodynamic Design of Bulbous Bows for Ship", Naval engineers Journal.
7. Tahara.Y , Wilson.R.V, Carrica.P.M (2006), "RANS simulation of a container ship using a single phase level-set method with overset grids and the prognosis for extension to a self-propulsion simulator"
8. Edward V. Lewis (1988), "Resistance Propulsion & Vibration", Principles of Naval Architecture Volume II.
9. Bertram.V (2000), "Practical ship hydrodynamics"