A Computational Method for Determination of Open Water Performance of a Marine Propeller

Senthil Prakash.M.N, PhD.
Associate Professor
Cochin University of Science And Technology
Kochi-36

Deepthi R Nath
MTech (CASAD)
Cochin University of Science And Technology
Kochi-36

ABSTRACT

Computational fluid dynamics (CFD), to which research efforts have been focused in the recent past is one of the most effective techniques of analysis of any fluid flow problems. Considerable part of towing tank tests for analysis of propeller characteristics are now being replaced by the CFD simulation on the computers. Here a four bladed Wageningen B screw series propeller has been designed for a specific vessel condition and numerically modelled and analysed using unstructured mesh in the flow domain. Validation of the numerical modelling of open water charectricts of screw propeller was achieved by comparing the results from the numerical analysis with the regression analysis based experimental series data.

Keywords
Marine Propellers, Computational Fluid Dynamics

1. INTRODUCTION

Ships are predominantly powered by means of marine screw propellers. The propellers when working at the aft of a ship they operate in a heterogeneous inflow field because of the wake created by the ship hull. Also the effect of suction produced by a working propeller affect the flow over the ship hull.Owing to these mutual interactive effects the analysis of the dynamics of flow is a complex and a difficult process. Recent modelling efforts have shown that CFD can provide valuable insight into the flow field generated by a propeller, including the forces and moments due to the rotating blades.

Research is going on for numerical prediction of performance of propellers by numerical methods. Some of them are listed in the referances [1]-[10]. Paweł Dymarski [5] has performed computations of the propeller open water characteristics using the SOLAGA computer program for predictions of the cavitation phenomenon. Senthil Prakash, M. N. and V.A.Subramanian [3], [4] and [6] has predicted the hull propeller interaction effects using RANSE and a potential flow simulation programme.Jang and Kinoshita [7] and Yoshihisa Takekoshi et al [8] have tried an optimization method on propeller blade. V.Anantha Subramanian and Senthil Prakash.M.N [2] has implemented a scheme of optimization for propeller by coupled VLM and RANS solver method

The simulation here is for the open water charactrestics of a B-Series propeller using the RANSE solver . The challenge in simulating the flow over propeller is in descretizing the flow domain, handling large number of cells in the flow domain and incorporating a rotating propeller for simulation.

2. PROPELLER IN OPEN WATER

A propeller is normally fitted to the stern of a ship where it operates in water that has been disturbed by the ship as it moves ahead. The performance of the propeller is thus affected by the ship to which it is fitted. Hence in order to determine the intrinsic performance characteristics of a propeller, unaffected by the ship to which it is fitted, it is necessary to make the propeller operate in open water. The performance characteristics of a propeller usually refer to the variation of its thrust, torque and efficiency with speed of advance and revolution rate in open water. In order to determine the open water characteristics of the propeller, experiments are performed with models of the propeller which is towed in the towing tank fixing the revolution rate and changing the towing speed. The thrust and torque produced by the propeller are measured. The non-dimensional thrust K_T and torque K_Q (which is magnified 10 times as $10K_Q$ to be plotted in the same graph) along with open water efficiency η_O are plotted as a function of advance coefficient J. The expressions for K_T, K_Q, η_o and J are given in equations 1-4

$$J = \frac{V_A}{nD} \tag{1}$$

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

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

$$\eta_o = \frac{K_T}{K_Q}\frac{J}{2\pi} \tag{4}$$

In the present study the designed propeller have been analysed using Computational Fluid Dynamics (CFD) for its open water characteristics instead of determining it experimentally.

3. PARTICULARS OF THE PROPELLER SELECTED FOR ANALYSIS

The design of a marine propeller is normally based upon charts giving the results of open water tests on a series of model propellers. These cover variations in a number of design parameters such as pitch ratio, blade area, and number of blades and section shapes. Many such series have been tested since the original work of Froude and Taylor. The most extensive is that of Netherlands Ship Model Basin (NSMB) at Wagenengen called the A and B series, run at various times between 1937 and 1964. It covers propellers having from 2 to 7 blades and wide range of blade area ratio.

In the B series of 4 bladed types, wider tips and the section shapes were aerofoil from root to 0.7 radius and circular back from the 0.8 radius to tip. This ensures reduced cavitation. In this a typical B series propeller is selected for the determination of characteristics performance. Diameter = 2m, Pitch = 1.6m, AE / AO = 0.55, Number of blades = 4, Shaft power = 700 Hp, RPM =257, Velocity of ship = 12 knots, Advance velocity = 9.48 knots

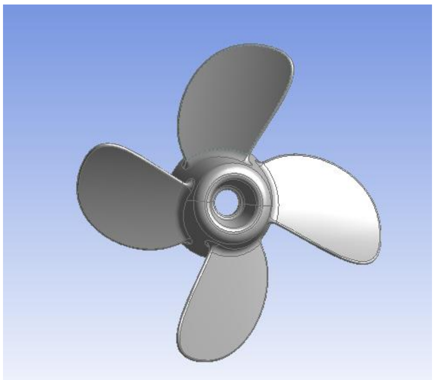

Fig.1.The B-Series Propeller (B4-55) designed for the above ship conditions

4. NUMERICAL ANALYSIS OF PROPELLER

Numerical studies have been carried out using computational fluid dynamics to obtain the open water characteristics of propellers as well as the distribution of pressure on the blade surfaces. The flow around the propeller is complex due to its complex geometry and the combined rotation and advancement into water. The flow around the propeller is governed by the following governing equations and the rotation of the propeller is incorporated by rotating frame of referance method available in CFD packages. Francesco [1] used a used a BEM technique for analysis with the assumption of inviscid flow..Here in CFD a FVM technique is used for simulations for the viscous flow

4.1 Governing Equations

The general conservative form of the Navier – Stokes equation is given below.

Continuity equation,

$$\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x_i}(\rho u_i) = 0 \qquad (5)$$

where ρ = density, u_i is the velocity component in the ith direction (i=1,2,3). The density is constant in case of incompressible flows. Since the propeller flow has been considered as steady and incompressible, the continuity equation gets modified as,

$$\frac{\partial}{\partial x_i}(\rho u_i) = 0 \qquad (6)$$

Momentum equation,

$$\frac{\partial}{\partial t}(\rho u_i) + \frac{\partial}{\partial x_j}(\rho u_i u_j) = -\frac{\partial p}{\partial x_i} + \frac{\partial \tau_{ij}}{\partial x_j} + \rho g_i + F_i \qquad (7)$$

where τ_{ij} is the Reynolds stress tensor given by

$$\tau_{ij} = [\mu(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i})] - \frac{2}{3}\mu\frac{\partial u_l}{\partial x_l}\delta_{ij}, \qquad (8)$$

p = static pressure,g_i = gravitational acceleration in the ith direction , F_i = external body forces in the ith direction and δ_{ij} is the Kroneker delta and is equal to unity when $i=j$; and zero when $i \neq j$.The Reynolds-Averaged form of the above momentum equation including the turbulent shear stresses is given by

$$\frac{\partial}{\partial t}(\rho U_i) + \frac{\partial}{\partial x_j}(\rho U_i U_j) = \frac{\partial}{\partial x_j}\left[\mu\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right) - \left(\frac{2}{3}\mu\frac{\partial u_l}{\partial x_l}\right)\right] - \frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j}\left(-\rho\overline{u_i' u_j'}\right)$$

. ……(9)

where u_i' is the instantaneous velocity component (i = 1,2 3).

For the closure of the RANSE equation a a turbulance model is used (Standard k-ε model)

4.2 Domain Specifications

.

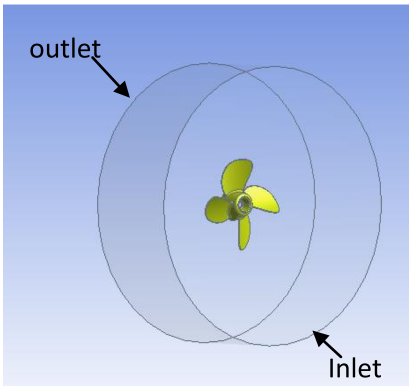

Fig.2 Propeller in the domain

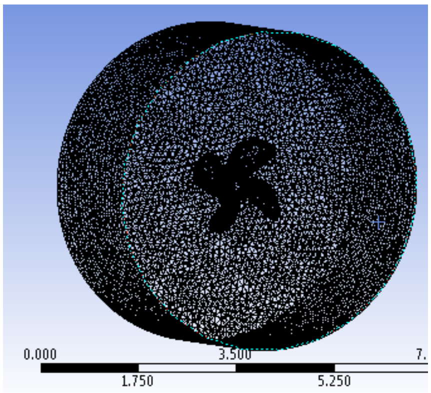

Fig.3 The propeller domain descretized with unstructured mesh

For flow simulation the propeller is considered to be enclosed in a cylindrical fluid domain. The inlet is at 0.5D upstream, outlet at 0.5D downstream, solid surfaces on the blade and the hub are centred at the coordinate system origin and are aligned with the inflow. The outer boundary is at 2D from the hub axis

Shin [9] and [10] has tried unstructured meshes for the domine descretization and applied navier-strokes equation based solver for propeller cavitation study. In the present investigation here the flow domine discussed above is descretized with unstructured mesh and a RANSE solver is used for flow simulation Fig.2.shows the domain with the boundary conditions and fig.3 shows the domain descretized with unstructured meshes

4.3 Solver Settings

Of the several options available, the selection of solver parameters for the propeller simulations are made partly from past experience, and partly from trial and error evaluations of various solver-setting combinations.

4.3.1 Boundary conditions

The flow in the given problem is controlled by the boundary conditions pertaining to the wall, velocity inlet, and outflow imposed on the domain.

The velocity inlet boundary condition is used to define the flow velocity along with all relevant scalar properties of the flow at the inlet. Velocity magnitude normal to the boundary is incorporated at the inlet.

The outflow boundary condition is used for the outlet. Outflow boundary conditions are used to model flow exits where the details of the flow velocity and pressure are not known prior to solving the flow problem.Conditions at outflow boundaries are not defined but extrapolated from the interior and hence have no impact on the upstream flow. The outflow velocity and pressure are updated in a manner that is consistent with a fully-developed flow assumption.

In any flow, Reynolds number of the flow becomes very low and turbulent fluctuations are damped considerably, near the walls.

The laminar viscosity starts to play a significant role. In the present case no slip condition is assumed at the blade

surface and allows slip near the lateral surface.The no slip condition assigned on the propeller walls permits calculation of drag and lift forces on that surface. In a propeller it is the lift force which provides the thrust and hence the thrust produced by the propeller can be extracted.

The lateral surface, even if is assumed at a distance of 2D there is no such boundary existing in a physical flow situation.Thus that boundary does not have any influence on the propeller action. This is attained by providing a boundary condition of wall which allows slip near the lateral surface.

4.3.2 Solver parameters

Solver parameter settings for the propeller open water simulations were made with judicious combination of the recommendations in the FLUENT literature, with trial and error evaluations of various solver settings. Combining these options with the domain dependence and grid dependence studies is a vital though time consuming initial effort before any propeller geometry can be investigated for performance. The final solver parameters including physical constants are shown below.

Table.1. Solver parameters

Parameters	**Settings**
Solver	Pressure based, transient
Velocity formulation	Absolute
Turbulence model	Standard k-ε
Water density	998.2kg/m^3
Water viscosity	0.001003kg/m-s
Pressure discretization	Body Force Weighted
Momentum discretization	First Order Upwind
Turbulent kinetic energy discretization	First Order Upwind
Turbulence dissipation rate	First Order Upwind
Pressure-velocity coupling	PISO

Table. 2. Boundary Conditions

Boundary	**Boundary conditions**
Blade surface	Wall (no slip)
Lateral surface of cylinderical domain	Wall (allows slip)
Inlet	Velocity Inlet : Velocity at advance speed
Out let	Outflow

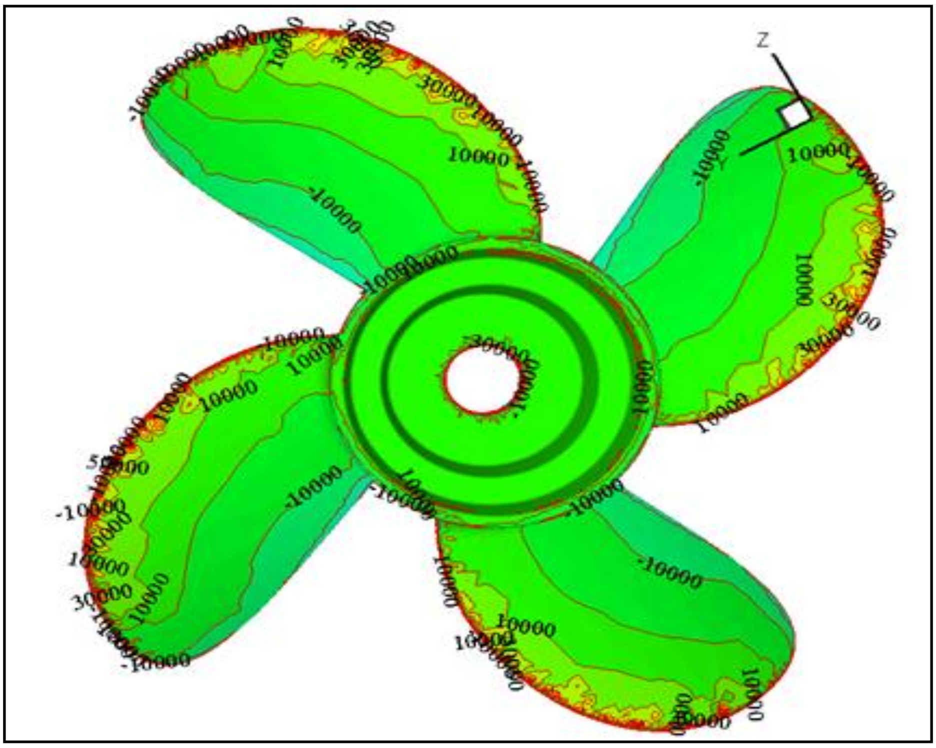

Fig.5. Pressure contours on the face of the propeller blades

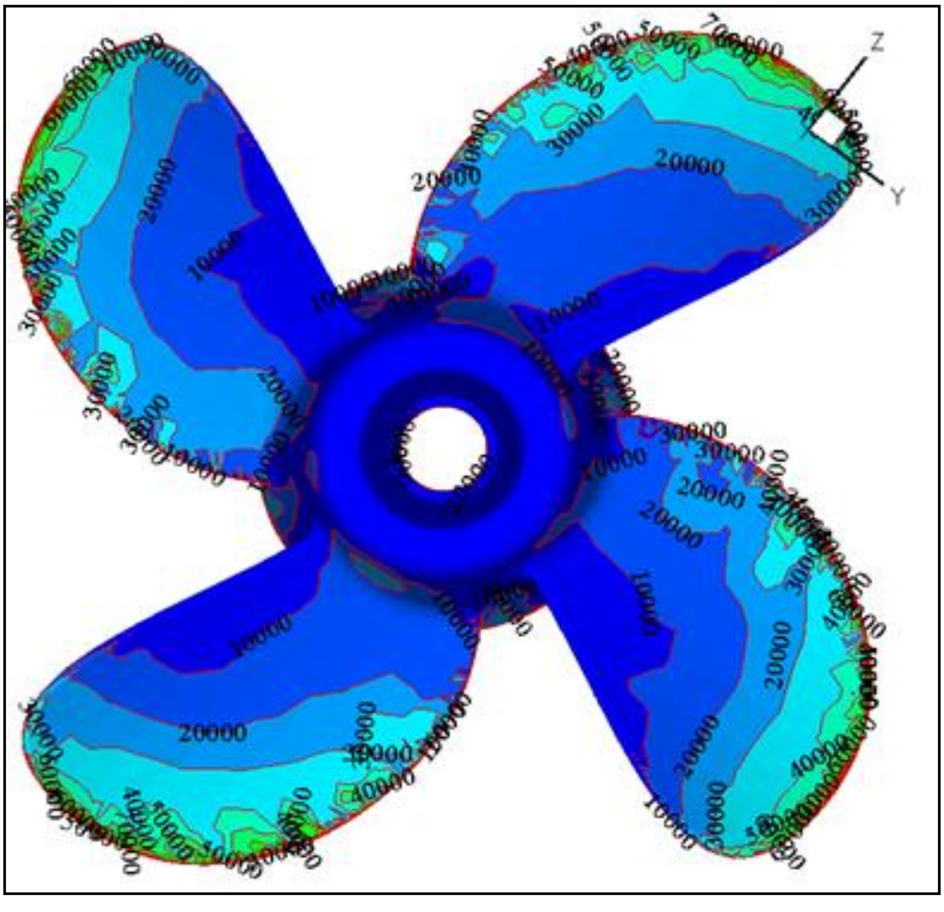

Fig.4. Pressure contours on the back of the propeller blades

5. RESULTS AND DISCUSSION

It can be observed from the figure that, there is a pressure difference between the propeller face and back region. The pressure difference between the face and back of the propeller results in the thrust. The thrust obtained from the pressure integration is 44.9KN, which is lesser in value as calculated by the methodical series data, which is 56 KN. This may be because of the unstructured meshing and due to the unsteady flow. Fig.4 shows the contours of hydrodynamic pressure (Δp) on the back of the propeller. As expected on the back of the propeller blade in ahead condition, the Δp is mainly negative, indicating that the pressure on the back of the propeller (Po). Fig.5 shows the contours of hydrodynamic pressure (Δp) on the face of the propeller. Again the contours are as expected with Δp being positive, indicating that the total pressure is greater than the ambient pressure on the face of the propeller.

Table 4.2 CFD results compared with the available experimental (regression analysis) results.

	Advance coefficient (J)	Thrust (KN)	Thrust coefficient (KT)	Torque coefficient (10KQ)	Efficiency(η)
CFD	0.567	44.9	0.186	0.176	0.953
propeller chart	0.569	56	0.1416	0.201	0.623

6. CONCLUSION

A Wagneignen B series propeller (4 bladed with expanded blade area ratio of 0.55 and pitch ratio 0.8) was successfully modelled and an unstructured mesh was applied to the model using CFD package ANSYS FLUENT. Open water test was simulated using FLUENT at velocity of advance 4.86 m/s. Fig.4 shows the contours of hydrodynamic pressure (Δp) on the back of the propeller. As expected on the back of the propeller blade in ahead condition, the Δp is mainly negative, indicating that the pressure on the back of the propeller (Po).Fig. 5 shows the contours of hydrodynamic pressure (Δp) on the face of the propeller. Again the contours are as expected with Δp being positive, indicating that the total pressure is greater than the ambient pressure on the face of the propeller.

The thrust has obtained from ANSYS FLUENT, advance coefficient of 0.567 was 44.9 KN and the corresponding thrust coefficient is 0.186, the thrust coefficient was evaluated using regression equation for the Wagneignen B series 4 bladed propeller, which yielded a vlue of thrust coefficient as 0.141. the percentage difference is found to be 30.1%.

Similarly the CFD has given a torque coefficient of 0.176, the corresponding value of torque coefficient as obtained from regression equation is 0.201, the percentage difference being 12.5%. the high percentage difference between the CFD values and the experiment based regression equation can be partly attributed to the fact that the CFD simulation was carried out using unstructured mesh which considerably reduces the accuracy of the results.

The propeller was modelled with a hole through the hub, but it was subsequently learned that this hole will be occupied by the propeller shaft. The presence of the hole in the propeller model used for simulation also can alter the flow through the propeller and may have contributed to the difference between the CFD results and the regression result.

It can be concluded that CFD can be successfully used to simulate the open water test of a propeller and to obtain thrust, torque and open water efficiency, better accuracy can be achieved by properly modelling the hub and also using a structured mesh.

7. REFERENCES

[1] Francesco Salvatore, Luca Greco, Danilo Calcagni Computational analysis of marine propeller performance and cavitation by using an inviscid-flow BEM model, Second International Symposium on Marine Propulsors smp'11, Hamburg, Germany, June 2011 Workshop

[2] V.Anantha Subramanian and Senthil Prakash.M.N, optimization of propeller by coupled VLM and RANS solver method**,** *Seventh International Conference On High-Performance Marine Vehicles Melbourne, Florida, USA 13-15 October 2010*

[3] Senthil Prakash, M. N. and V.A.Subramanian, **"**Simulation of propeller- hull interaction using RANSE solver", *International Journal of Ocean and Climate Systems(2009),* Vol.1,No.3,4.pp.189-203

[3] Senthil Prakash, M. N. and V.A.Subramanian, "Body force based simulation of propeller hull interaction", *Proceedings of 3rd International Conference in Ocean\ Engineering*, IIT Madras, Chennai, India, 1-5th February 2009.

[5] Paweł Dymarski 2008 Computations of the propeller open water characteristics using the SOLAGA computer program. Predictions of the cavitation phenomenon , Archives of Civil and Mechanical Engineering, Volume 8, Issue 1, 2008

[6] Senthil Prakash, M. N. and V.A.Subramanian, **"**Simulation of propeller-ship hull interaction using an integrated VLM/RANSE solver modeling*", 6th International Conference on high performance marine vehicles,* Naples, Italy, 18-19th September 2008.

[7] T S Jang and T Kinoshita, 2008. An optimization theory and its application to CFD based design for marine propeller behind ship. Journal of the Institute of Industrial Science, University of Tokyo, Volume 52, No 8, 2008.

[8] Yoshihisa Takekoshi et al, 2005. Study on the design of propeller blade sections using the optimization algorithm, Journal of marine science and technology, 10:70-81, 2005.

[9] Shin Hyung Rhee and Shitakumar Joshi, 2005. Computational validation for flow around a marine propeller using unstructured mesh based navier-strokes solver, JSME International Journal, Volume 48, No 3, 2005.

[10] Shin Hyung Rhee et al, 2005. Propeller cavitation study using an unstructured grid based navier-strokes solver. Journal of fluids engineering, Volume 127, September 2005.