JOURNAL OF MARITIME RESEARCH

Vol XIII. No. XI (2014) pp 71–76

ISSN: 1697-4040, www.jmr.unican.es

Numerical Investigation for CFD Simulation of Open Water Characteristics and Cavitation Inception of Marine Propeller Blade

M.P. Kishore[1,*], R.K. Behera[1]

ARTICLE INFO

Article history:
Received 17 February 2016;
in revised form 19 February 2016;
accepted 28 March 2016.

Keywords:
Cavitation, CFD, Fluent, Open Water Characteristics, Propeller.

ABSTRACT

With the recent advancements in underwater technology the need for improvement in operation and range of underwater vehicles to operate at abnormal speed increased timely. To achieve this need for improvisation of open water efficiency of the marine propeller blade is taken into consideration. The operational range of such marine propellers mainly depends on its open water characteristics prediction. This paper investigates about the methodology for determination of open water characteristics and cavitation inception of a marine propeller which are important for prediction of propeller performance. To predict, the open water characteristics a four bladed propeller is considered to compute the flow simulation, cavitation inception, velocity flow fields around the propeller blade in a uniform wake flow. Steady Reynolds-Averaged Navier-Stokes [RANS] simulations with k-omega ($k - \omega$) SST (shear stress transport) turbulence model and wall functions in combination with Multiple Reference Frames [MRF] were used for the simulations. The results obtained from the CFD simulations are validated with the earlier works.

1. Introduction

The increase in demand for high efficiency propeller makes difficult to avoid the occurrence of cavitation .Cavitation on marine propellers cause noise vibration thrust reduction and erosion. Practically the analysis of determining the cavitation inception for marine propellers depends on cavitation tunnel tests, empirical data and in viscid flow method. Performing a series of experiments of model test is costly and the later two methods have less accuracy. A moving ship experiences resisting forces produced from the water which must be overcome by thrust produced by means of some mechanism. In the earlier days consisted of manually operated oars which gave place in turn to sails and then mechanical devices such as jets, paddles wheels and propellers of many different forms came to existence. Propellers more particularly propeller blades are complex shapes which require the right hydrodynamic surfaces. Most of the cad tools which handle these complex shapes and surfaces. In the design stage propeller operating in underwater conditions depends upon its ability to determine its thrust and torque accurately. In general the efficiency of propulsion system mainly depends upon its propeller performance, force, torque efficiency. Its principal parameters to be determined. The hydrodynamic aspects including the thrust deduction, wake, characteristics of propeller are of importance. Owing to the fact that the analysis of dynamic of flow is complex and difficult process for prediction, recent simulations for these type of interactive effects has shown that CFD can provide valuable insight into the flow field generated by a propeller including forces and moments due to rotating blades. The simple method for assessment of marine propeller hydrodynamic performance is to graph propeller coefficient against advance coefficient (J). The classical blade element theory of propellers is used to determine the propeller characteristics and the force distribution acting on the propeller. The finite element method used to determine the resulting deformation of the propeller blades. Weick (F. E. Weick, 1930a)(F. E. Weick, 1930b) presented the results of wind tunnel tests carried out on various conventional metal-

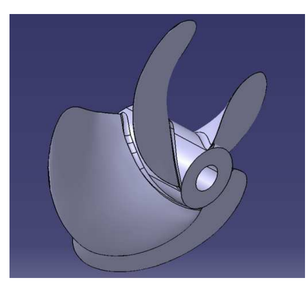
[1]Department of Mechanical Engineering, National Institute of Technology Rourkela Rourkela, INDIA.

*Corresponding Author: M.P. Kishore. E-mail address: kishorecadcam@gmail.com

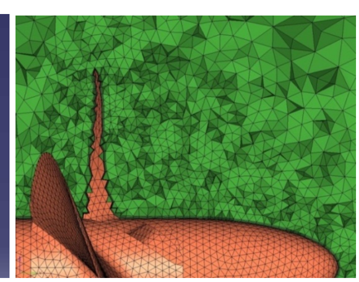
lic propellers. Pawel Dymarski (Paweł, 2008) has performed computations of the propeller open water characteristics using solaga computer programme for determination of cavitation phenomena. The fluid structure interaction analysis of flexible composite marine propellers subject to hydrodynamic and inertial loads was also presented to Young (Young, Y.L., 2008a). Young & Savander (Young, Y.L. and Savander, B.R., 2011) reported the numerical analysis of a large surface piercing propeller. Young (Young, Y.L., 2007) presented a coupled boundary element method BEM and finite element method (FEM) for the numerical analysis of flexible composite propellers in uniform flow and wake flow. This research is extended to the fluid structure interaction analysis of flexible, composite marine propellers subjected to hydrodynamic and inertial loads. The hydro dynamic blade, loads, deflections, stress distribution of flexible composite propellers can be predicted by method (Paweł, 2008). A coupled structural and fluid flow analysis was performed to assess the hydro elastic behavior of composite marine propeller. The forces acting on the blades and Stress strain relations are calculated by using analytical and theoretical experimental relations Sontvedt (Sontvedt, 1974) achieved using shell elements, the results for prediction of quasi static and dynamic stresses in marine propeller blades. (Chau, 2010),(Sontvedt, 1974) carried out a comprehensive work on the hydro elastic tailoring of the flexible Composite propeller. A coupled structural fluid flow analysis was performed by Blasques et al (Blasques et al., 2010) to evaluate the hydro elastic behavior of a composite marine propeller. Senthil, prakash, MN and VA Subramanian (Senthil Prakash, M. N. and V.A. Subramanian , 2009b), (Senthil Prakash, M. N. and V.A. Subramanian , 2009a), (V.Anantha Subramanian and Senthil Prakash. M.N, 2010) predicted the hull propeller interaction effects using Ranse and a potential flow simulation programme. Blade stress strain relation of the marine propeller was analyzed by Chau (Chau, 2010). Koronowicz (Koronowicz et al., 2009) presented the comprehensive program to account for the hull-propeller rudder system in the propeller design process. The program outcome includes the hydrodynamic performance cavitation effects blade strength and efficiency optimization. V. Anantha Subramanian and Senthil Prakash (Senthil Prakash, M. N. and V.A. Subramanian , 2009b) has implemented a scheme of optimization for propeller by coupled vlm and Rans solver method. A ship propeller design (SPD) software code was prepared by Ghassemi (Ghassemi, 2008) and employed for various propulsors such as propeller rudder system a highly skewed propeller (Salvatore et al., 2011), a contra rotating propeller and spp (Ghassemi, 2009b). This code employs the BEM including the boundary layer theory to determine the hydrodynamic analysis of marine propeller (Ghassemi and Ghadimi, 2007). Chattopadhayay (Chattopadhyay et al., 1995) used the classical blade element theory in an optimization procedure for improving the performance of a high speed rotor. For the purpose of structural analysis they used the finite element method. Chazly (N. M. Chazly., 1993) performed static and dynamic analysis of a wind turbine blade. For computation of stress deflection patterns and eigen values of metallic blade a triangular bending element was used. Bernard (Bernad, 2006) presented a numerical investigation of cavitating flows using the mixture model implemented in the fluent 6.2 commercial code. Sridhar (Sridhar et al., 2010) predicted the frictional resistance offered to a ship in motion using fluent 6.0 and these results are validated by experimental results. Chang (Blasques et al., 2010) applied finite element volume CFD method in conjunction with the standard K-e turbulence model to calculate the flow pattern and performance parameters of a DTNSRDC P4 119 marine propeller in a uniform flow. Sanchez-caja (Sanchez-Caja, 1998) has calculated open water flow patterns and performance coefficients for DTRC 4119 propeller using FINFLO code. The flow patterns were generally predicted with K-e turbulence model and suggested a better prediction of tip vortex flow which requires a more sophisticated turbulence model. Salvatore (Salvatore et al., 2011) performed computational analysis by using Insean-PFC propeller flow code developed by CNR Insean. Experiments are carried to know the open water characteristics evaluation of velocity field in the propeller wake and prediction of cavitation in uniform flow conditions.

2. Catia as Base Line Modeler for Geometric Modeling

The three dimensional modeling of propeller blade along with hub is done using CatiaV5R20. The point coordinates are converted to generate aero foil points. These aero foil points are plotted in three dimensional spaces joined by means of a smooth curve. They are rotated with respective pitch angle and wrapped around a cylindrical diameter to attain final sections .by using loft command these sections are joined to get final surface model, by using solid option surface model is converted to three dimensional models. The following figure shows the solid model of 4bladed propeller.



(a) Threedimensional model of propeller



(b) Prism layers around the Blades with close view

3. Domain Specifications and Grid Generation

The flow simulation of a propeller is conducted in a cylindrical domain filled with fluid. The inlet of the domain was considered at a distance of 3D from the mid chord of the root of the section an outlet is considered at a distance of 4D from same point of downstream from the axis of hub the domain was constructed with a diameter of 4D in radial direction. This peripheral plane is to referred as far field. For the computational analysis shafts and fairing caps are attached to the propeller hub.

The flow domain is discretized to convert the partial differential equations into series of algebraic equations. This process is called grid generation. To generate grid with tetrahedral elements commercially available grid generation code Hypermesh 11.0 solver is used. The total number of elements created is varied to 0.21million. The flow simulations of the propeller was done by using Fluent6.3. From the figure it is clear that denser mesh is near the propeller surface to capture the flow properties with significant quality.

4. Solver Parametres for Propeller

With the judicious combinations of the recommendations in the literature of fluent the solver settings for the propeller in open water simulations are made with trial and error evaluations of various configurations fluent 6.3 code is used to sole the three dimensional viscous incompressible flow simultaneously the parallel version computes the flow equations using multiple processors. The options of combining the grid and domain dependence studies becomes ital. while judging any propeller geometry for its performance. The final solver parameters are as shown.

4.1. Flow Solution and Solver Settings

The Fluent 6.3.26 code was used to solve the three dimensional viscous incompressible flow.

Table 1: Propeller Details and solver settings

Propeller	SDB
Principal Dimensions	Propeller Diameter $(D) = 0.205\ m$
Domain Size	Cylindrical Domain of Length 1.435 m ($7D$), dia 0,82 m ($4D$)
Mesh Link	5.5 lakh tetrahedral and prism cells
Pressure Link	SIMPLE
Pressure	Standard
Discretization Scheme for Convective fluxes and Turbulence Parametres	First Order Upwind
Turbulence Model	RNG $K = \varepsilon$
Near Wall Treatment	Standard Wall Functions
Solver	Steady
Operating Pressure	140 kpa
Vapour Pressure	5.0 Kpa

5. Results

The pressure difference over the suction and back side region of the propeller blade is shown in the figure. The difference of pressure on the face and back results in thrust. To develop propulsion factors thrust, torque, speed of rotation, speed of hull is to be determined either using propulsion test or analysis package. The propulsion factors represented in terms of non-dimensional coefficients, Thrust coefficient (K_T), Torque coefficient (K_Q), Advanced coefficient (J) are to be used for determination of open water Efficiency (η_0).These non-dimensional terms expressing the general performance characteristics are

Advanced coefficient:

$$(J) = \frac{V_a}{n \cdot D} \tag{1}$$

Thrust coefficient:

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

Torque coefficient:

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

Open water Efficiency:

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

The comparison of calculated and reference values of non-dimensional terms are plotted in the table.

Figure 1: Computation of K_T, K_Q and Open water efficiency (η_0) for present and validation with reference

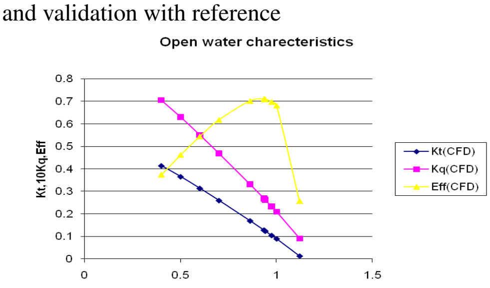
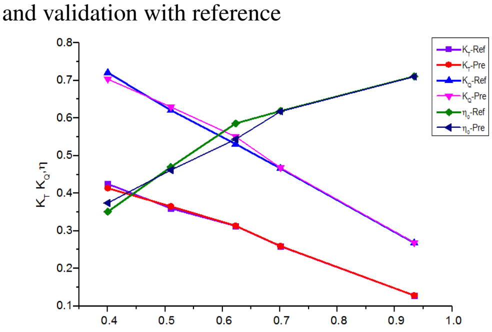


Figure 2: Computation of K_T, K_Q and Open water efficiency (η_0) for present and validation with reference



The above values of thrust coefficient (K_T), toque coefficient (K_Q) and open water efficiency (η_0) are plotted against

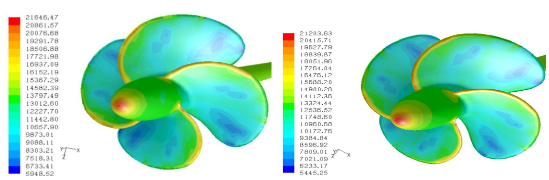
Table 2: Open water characteristics results obtained from CFD analysis

J	$V_a(m/s)$	N(Rpm)	T(N)	Q($N-m$)	K_T	$10K_q$	η_0
1.00	3.83	1120	55.21	2.642	0.0897	0.2094	0.6817
0.974	3.83	1150	67.80	3.085	0.1046	0.2324	0.6982
0.934	3.83	1200	90.51	3.882	0.1283	0.2685	0.7104
0.862	3.83	1300	140.14	5.615	0.1693	0.3309	0.7021
0.7	3.83	1600	325.90	12.05	0.2595	0.4680	0.6177
0,6	3,83	1800	535.91	19.28	0.3135	0.5504	0.5439
0.5	3.83	2200	899.32	31.77	0.3653	0.6297	0.4617
0.4	3.03	2800	1590.79	55.50	0.4136	0.7039	0.374

Table 3: Comparison of Thrust, Torque and Advanced coefficient values with Reference

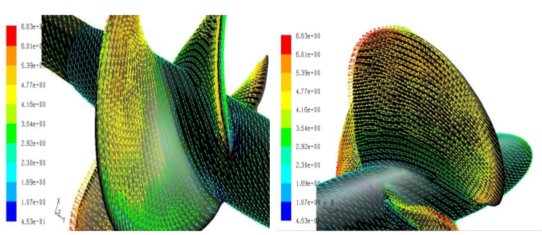
$V_a(mm/S)$	N(Rpm)	Advanced Coefficent (J)		Thrust T(N		Torque Q($N-m$)	
		Ref	Present	Ref	Present	Ref	Prensent
3.83	1200	0.934	0.934	90.51	90.51	3.88	3.882
3.83	1600	0.701	0.70	324.92	325.91	12.01	12.05
3.83	1800	0.623	0.60	498.17	535.91	17.29	19.28
3.83	2200	0.510	0.50	854.58	899.32	30.24	31,77
3.83	2800	0.400	0.40	1587.42	1590.79	55.38	55.50

Figure 3:



(a) Absolute pressure contour at N = 1120 rpm & J = 1.01

(b) Absolute pressure contours at N = 1150 rpm & J = 0.974



(c) Relative velocity vectors at blade surface for V_a = 3.83 m/s, and J = 0.934

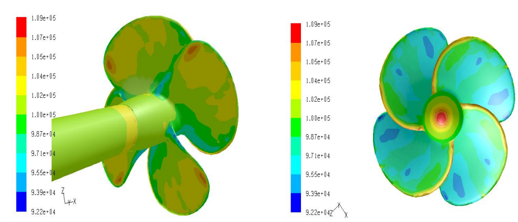
(d)

advance coefficients (J) to get the open water characteristics as shown in the figure 4.

CFD analysis for Rotational speed = 1100 rpm and Advanced velocity = 3.83 m/s

Relative Velocity distribution at N =1200 rpm and J = 0.9341 around the blade-surface is shown in fig. The figure shown above represents that there is no flow separation near the blade surface section, which was expected as the propeller was a well-designed standard one.

Figure 4: Absolute pressure on face and back of propeller at J = 0.9341 and N =1200 rpm

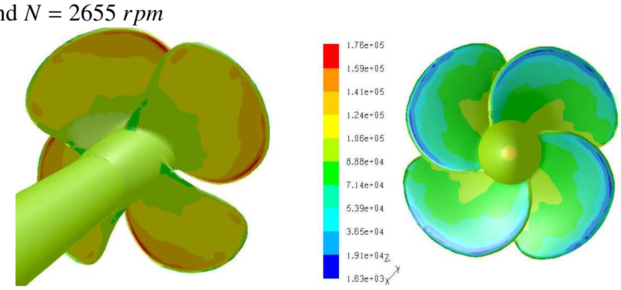


(a) (b)

6. Cavitation

The figure 5 shows relative velocity distribution at J = 0.4222 and N = 2655 rpm around the blade-surface.

Figure 5: Absolute pressure on face and back of propeller at J = 0.4222 and N = 2655 rpm



Contours and pressure plot in fig 5 shows that at J = 0.4222 minimum absolute pressure on blade is1630 Pa which is less than the vapour pressure 1720 Pascal. This is the stage at which cavitation is just started which is the point of Cavitation Inception. Non dimensional parameter Cavitation number (σ) at this instant is 13.6. Type of cavitation observed from pressure con-

Table 4: Comparision of Predicted and reference (29) values of K_T, K_Q, η_0

Advance Coefficient (\$j)		Thrust Coefficient (K_T)		Torque Coefficent ($10K_Q$)		Efficiency (η_0)	
Ref	Present	Ref	Prensent	Ref	Present	Ref	Present
0.934	0.934	0.128	0.1283	0.268	0.2682	0.711	0.7104
0.701	0.70	0.259	0.2595	0.466	0.468	0.619	0.6177
0.623	0.60	0.313	0.3135	0.531	0.5504	0.586	0.5439
0.510	0.50	0.360	0.3653	0.621	0.6297	0.470	0.4617
0.400	0.40	0.425	0.4136	.721	0.7039	0.351	0.374

tours may be predicted as tip vortex cavitation along the tip of leading edge with the help of Pressure contours shown in fig **??**

Figure 6: Absolute pressure distribution graph at $\frac{r}{R_p}$ = 0.73 and N = 2655 rpm

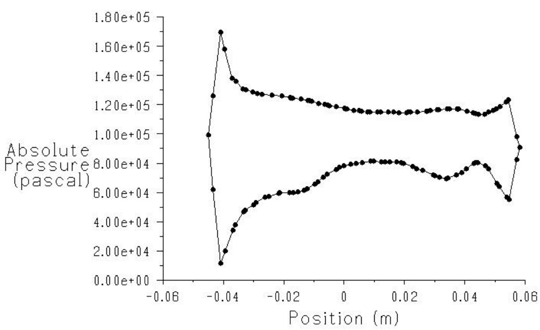


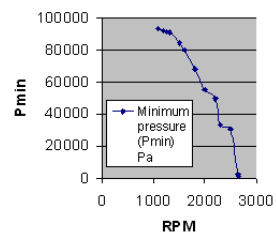
Table 5: Relation for RPM and minimum pressure

Speed (m)	Advance Coefficient (J)	Minimum Pressure ($Pmin$) Pa
1080	1.037	93500
1200	0.9341	92200
1260	0.8896	91500
1320	0.8492	90800
1500	0.7473	84600
1600	0.7	79700
1800	0.622	68400
2000	0.5604	55500
2200	0.5095	50000
2300	0.4873	33000
2500	0.4483	30600
2640	0.4311	2640
2655	0.4222	1630

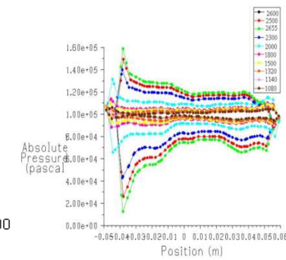
Comparisons of pressure distribution around the propeller's blade hydrofoil at various rpm is shown in fig 6

7. Conclusion

In this study the open water characteristics are predicted computationally on the basis of a validated small sized propeller where the delivered power (PD), the advanced coefficient (V_a), the propeller revolution (N) are known.



(a) Minimum pressure vs. RPM



(b) Pressure plot for decreasing minimum pressure towards cavitation inception

The following conclusions are arrived from this comparative analysis

1. Maximum open water efficiency for computational result is 71.04% which is the instant exactly at operating conditions J = 0.934, V_a = 3.83 and N = 1200 rpm show as the computational results are up to the mark.
2. With variation of thrust coefficient from 0.0897 to 0.3135 and Torque coefficient from 0.2094 to 0.5504 the advanced coefficient (J) varied from 1.00 to 0.6. Computational results obtained from fluent software are in good agreement with the reference [29]. Further experiments can be done to validate the numerical results obtained for better reliability.
3. More amount of research can be conducted in relation to the refinement of mesh and more particularly an improved way of combining the blade and hub of propeller inside the block. For attaining close form of results tetrahedral element mesh is replaced with hexahedral elements.
4. Further trials can be conducted over varying advanced velocities and rotational speed in order to have conclusive results in comparison with experimental results. This additional analysis provides more amount of information for enhancement of open water efficiency and distribution of pressure over the blades.

References

Bernad, S., (2006). Numerical analysis of the cavitating flows. Center of Advanced Research in Engineering Sciences, Romania Academy. Timisoara Branch, Romania.

Blasques, J., Berggren, C., Andersen, P., (2010). Hydro-elastic analysis and optimization of a composite marine propeller. Mar Structs Vol 23 (No. 1), pp 22–38.

Chang, B., (1998). Application of cfd to p4119 propeller. 22nd, ITTC Propeller RANS/Panel Method Workshop, France.

Chattopadhyay, T., McCarthy, R., Seeley C., (1995). Decomposition based optimization procedure for high speed prop-rotors using composite tailoring. J. Aircraft Vol 32 (No. 5), pp 1026 –1033.

Chau, T., (2010). 2-d versus 3-d stress analysis of a marine propeller blade. Zeszyty Naukowe Akademii Morskiej w Gdyni No. 64.

F. E. Weick, (1930). Full scale wind tunnel tests with a series of propellers of different diameters on a single fuselage. NACA Rep. 339.

F. E. Weick, (1930). Scale wind tunnel tests on several metal propellers having different blade forms. NACA Rep. 340.

Ghassemi, H., (2008). Hydrodynamic characteristics of the surface-piercing propellers for the planing craft. J. Mar Sci Applic Vol 7, pp 147–156.

Ghassemi, H., (2009). The effect of wake flow and skew angle on the ship propeller performance. Scientia Iranica Vol 16 (No.2), pp 149–158.

Ghassemi, H., (2009). Hydrodynamic performance of coaxialcontra-rotating propeller (ccrp) for large ships. Pol Marit Res Vol 16 (No. 59), pp 22–28.

Ghassemi, H., Ghadimi, P., (2007). Computational hydrodynamic analysis of the propeller–rudder and the azipod systems. Ocean Engg Vol 34, pp 117–130.

Koronowicz, T., Krzemianowski, Z., Tuszkowska, T., Szantyr, J., (2009). A complete design of ship propellers using the new computer system. Polish Maritime Research Vol 16 No.1 (59), pp 29–34.

Lin, H., W., L., Kuo, Y., (2010). Effect of stacking sequence on nonlinear hydroelastic behaviour of composite propeller. Journal of Mechanics Vol 26 (No. 3).

Mulcahy, N., Prusty, B., Gardiner, C., (2010). Hydroelastic tailoring of flexible composite propellers. Ships and Offshore Structures Vol 5 (No.4), pp 359–370.

N. M. Chazly., (1993). Static and dynamic analysis of win turbine blades using the finite element method,. Comput . Struct Vol 48 (No.2), pp 273–290.

Paweł, D., (2008). Computations of the propeller open water characteristics using the solaga computer program. predictions of the cavitation phenomenon. Archives of Civil and Mechanical Engineering Vol 8, Issue 1.

Ravinder reddy, P., S., (2014). Design of hybrid composite marine propeller for improved cavitation. International Journal of innovative research in Technology and Science ISSN 2321-1156.

Salvatore, F., Greco, L., Calcagni, D., (2011). Computational analysis of marine propeller performance and cavitation by using an in viscid-flow bem model. Second International Symposium on Marine Propulsors Hamburg, Germany.

Sanchez-Caja, A., (1998). P4119 rans calculations at vtt. 22nd, ITTC Propeller RANS/Panel Method Workshop, France.

Senthil Prakash, M. N., V.A. Subramanian , (2009). Body force based simulation of propeller hull interaction. Proceedings of 3rd International Conference in Ocean Engineering, IIT Madras, Chennai, India.

Senthil Prakash, M. N., V.A. Subramanian , (2009). Simulation of propeller-hull interaction using ranse solver. International Journal of Ocean and Climate Systems Vol 1 (No3,4), pp 189–203.

Solomon raj, S., Ravinder reddy, P. , (2014). Bend twist coupling and its effect on cavitation inception of composite marine propeller. International Journal of Mechanical engineering and technology (IJMET) Vol 5, Issue 9, pp 306–314.

Sontvedt, T., (1974). Propeller blade stress application of finite element methods. Computers and Structures Vol 4, pp 193–204.

Sridhar, D., Bhanuprakash, T. V. K., Das, H. N., (2010). Frictional resistance calculations on a ship using cfd. Int. J. of Computer Applications Vol 11 (No.5), pp 24–31.

V.Anantha Subramanian, Senthil Prakash. M.N, (2010). Optimization of propeller by coupled vlm and rans solver method. Seventh International Conference On High-Performance Marine Vehicles Melbourne, Florida, USA.

Young, Y.L., (2007). Hydro-elastic behavior of the flexible composite propeller in wake inflow. 16conf. on composite materials, Kyoto, Japan.

Young, Y.L., (2008). Fluid–structure interaction analysis of flexible composite marine propellers. J Fluids Structs 24, pp 799–818.

Young, Y.L., (2008). Fluid-structure interaction analysis of flexible composite marine propellers. Journal of Fluids and Structures Vol 24, pp 799–818.

Young, Y.L., Savander, B.R., (2011). Numerical analysis of large-scale surface-piercing propellers. Ocean Engg Vol 38 (No 13), pp 1368–1381.