Hydrodynamic characteristics of marine composite propeller blade using a numerical approach

M L P Kishore[1], Vijay K Singh[2], R K Behra[3], Chandra S Saran[4], Manikant Paswan[5], Kapil Kumar[6]
[1]Department of Mechanical Engineering, ICFAI University, Telangana, India
[2,6]National Project Implementation Unit-Ministry of Human Resource and Development, MMMUT, Uttar Pradesh, India
[3]Department of Mechanical Engineering, NIT Rourkela, Odisha, India
[4]Department of Mechanical Engineering, Government College of Engineering, Odisha, India
[5]Department of Mechanical Engineering, NIT Jamshedpur, Jharkhand, India

Article Info

Article history:

Received May 3, 2020
Revised Jul 6, 2020
Accepted Sep 29, 2020

Keywords:

B-series composite propeller blade
Hydrodynamic characteristics
Open water efficiency

ABSTRACT

The aim of the present research work is to investigate the hydrodynamic characteristics (pressure distribution, rotational speed, thrust and torque) of the conventional B-series composite propeller blade. The open water efficiency for the scaled model of composite propeller blade is computed using computational fluid dynamics (CFD) fluent simulation tool. The obtained numerical results show that the propeller will operate at optimum efficiency for the given speed condition and perform with reduced efficiency at other operational speeds. The computed responses are also validated with the standard B-series data which verifies the accuracy and robustness of the present numerical approach in analyzing the performance characteristics of propellers. The deviation in solution ranges from 5 to 15% in the case of thrust, 10-20% in case of torque. Pressure estimation is usually quite accurate with a 5-8% variation. The tabular data of pressure distribution over the propeller blade may be used for further structural analysis.

Corresponding Author:

Vijay K Singh
National Project Implementation Unit-Ministry of Human Resource and Development
MMMUT
Gorakhpur-273010, Uttar Pradesh, India
Email: vijaynitr12@gmail.com

1. INTRODUCTION

Since 19th century, screw propellers have secured a position in the ship hydrodynamics as the best propulsion system despite of several developments in the propeller design and propulsion system due to its suitable use and higher efficiency over a long period of time. With the rapid advancements in technology, the determination of principal dimensions like ship size, speed for the design of ship using numerical method-based tools has gained much importance. These numerical applied computational methods have gained the closed form of solutions with analytical or experimental methods. The use of Wageningen B-series as base series for marine propeller blades proved desirable for various applications of ocean engineering. Although much research has been done on the optimal design and applications of such series, the baseline lies in determining the open water characteristics of these blades to assess the suitability in underwater conditions. A normal procedure is to design a propeller is based on the diagrams obtained from the open water test of the model series. The marine propeller design and determination of optimum diameter can be made using the data from the polynomials of $K_{T,} K_{Q,} B_{P,} - \delta, B_{P,} - U$ diagrams. The alternative is to use mathematical methods like lifting line, lifting surface, Boundary element method, vortex lattice methods. With huge developments in the technology of computers, great improvements occurred in the propeller's design followed by its analysis. The application of computers reduced the time and effort involved to carry out number of experiments in towing tanks by designing the performance of propellers with a just few design parameters and considered to be one of the most effective techniques for analysis of fluid flow parameters. The design of marine propeller was done [1], systematic propeller series and implemented numerical relations in the design of propeller blade (B-series) under various loading conditions. A three-dimensional hydrofoil in a water tunnel has been investigated with respect to its' fluid-structure interaction (FSI), Shiliang HU et al [2] using the Finite volume method (FVM) and Finite element method (FEM). The propeller efficiency (open water and body) for single and twin-screw ships was computed [3], using several advanced coefficients definitions. The comprehensive description of the state of art can be found [4], where the flow across a propeller examined experimentally in open water channels keeping the flow conditions uniform.

The flow field and physical parameters subjected to a propeller simultaneously was analysed [5] using computational fluid dynamics (CFD) technology. While dealing with the turbulent shock wave and break wave to take care nonlinearity, they applied the Navier stokes equation has been applied to the solver. The thrust and torque performance of the marine propeller blade at different rotational speed has been estimated using CFD and found a good relevance with the experimental results reported [6, 7]. A series of marine propeller have tested under open water experimentation and the obtained hydrodynamic characteristics was compared with that of the numerical results [8], A performance test was carried out [9] for 79 propellers in the diameter range of 9 to 11 inch along with the measurement of thrust as well as torque in the propeller speed range 1500-7500 rpm with respect to its' diameter. The complementation of combined blade–momentum element-theory for the propellers which are light/moderately loaded was illustrated [10]. The hydrodynamic characteristics for the propellers (conventional metallic) of various sizes was determined [11, 12] using cavitation and tunnel test.

Most of these research work had been carried out either by analytically or numerically in viscid or fully viscous/boundary layer technique [13]. With the effects of turbulence flow taking into consideration and flow field estimation with no-slip boundary conditions are out of important RANS (Reynolds average Navier stokes equation) with appropriate turbulence model is used to predict the flow characteristics of propeller blade according to [14]. A CFD based lifting line surface theory was used [15, 16] for visualizing the flow pattern around the merchant's ship propeller. The viscous equation solution for the flow pattern was implemented [17]. A ship propeller design (SPD) computer code developed and employed for many propulsors (propeller rudder system and contra rotating propeller [18-22]. The hydrodynamic analysis of marine propeller was performed using this SPD code based on boundary layer theory. A coupled boundary element method along with the finite element method was applied to determine the numerical solution for the flexible propellers (composite) [23]. A numerical analysis was carried out [24] in the case of propellers of large-scale surface. The open water characteristics was computed along with the analysis of cavitation problem [25] using computer code (solaga). Senthil et al [26] obtained the open water characteristics of propeller (B-series) using CFD simulation. Later on, the CFD analysis of propeller (contra-rotating) was performed by Das et al [27]. Das and Jaykumar [28] predicted the hydrodynamic characteristics of a contra-rotating propeller by validating the computed results with the experimental data. [29] Studied an autonomous underwater vehicle in the case of hull propulsor interaction by performing the computational analysis and experimental validation. Krishna et al. [30] presented the acoustic characterization of a Benchmark Marine propeller using CFD. The numerical prediction was made [31] for the forces acting on propellers (podded and open), working under the abnormal flow situations. Subramanian and Senthil [32] implemented a scheme for the optimization of the propeller by coupled LM and RANS solver methods. Application of CFD for the prediction of performance characteristics in marine propeller can be found in Kimura et al. [33], Takshi and Jun [34], Choong et al. [35]. A mechanical prototype of (OWC) Oscillating conversion Chamber device was built and studied using CFD simulator [36]. A comprehensive study on bend-twist coupling of composite propeller blades in order to improve the hydrodynamic efficiency was presented by Das and Kapuria [37].

The objective of the present study is to perform the Computational fluid dynamics simulation for the flow around the B-series Wageningen propeller using an unstructured mesh-based Reynolds-averaged Navier stokes equation method. The computed results are validated with the standard B-series data in order to test the efficacy of the present method. A moving reference frame approach is selected using the K-ε model and employed for turbulence closure. The global quantities such as thrust and torque coefficients along with open water efficiency (K_T, K_Q, and η_o) are determined in addition to pressure contours on the blade surfaces. The deviation in solution has been found from 5 to 15% in the case of thrust and 10-20% in the case of torque. Pressure estimation is usually quite accurate with 5-8% variation.

2. MATHEMATICAL FORMULATION
2.1. Governing equations

For the practical design consideration and performance analysis point of view empirical relations along with diagrams are presented for B-series propeller. The parameters contained for design of a propeller blade include the power delivered (PD), the number of blades blade area ratio, advanced speed, number of propeller revolutions pitch ratio and the diameter D, by systematically changing the advanced speed or propeller revolutions the performance characteristics are investigated among probable solutions. The variable parameters representing this series are the propeller blade number, the blade area ratio and the pitch ratio.

$$K_Q = \sum_{n=1}^{47} C_n (J)^{sn} (\frac{P}{D})^{\text{tn}} (\frac{A_E}{A_o})^{un} (Z)^{vn} \tag{1}$$

$$K_T = \sum_{n=1}^{39} C_n (J)^{sn} (\frac{P}{D})^{\text{tn}} (\frac{A_E}{A_o})^{un} (Z)^{vn} \tag{2}$$

The dimensionless characteristics are expressed as

$$J = \frac{V_a}{nD} \tag{3}$$

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

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

$$\eta_o = \frac{J}{2\pi} * \frac{K_T}{K_Q} \tag{6}$$

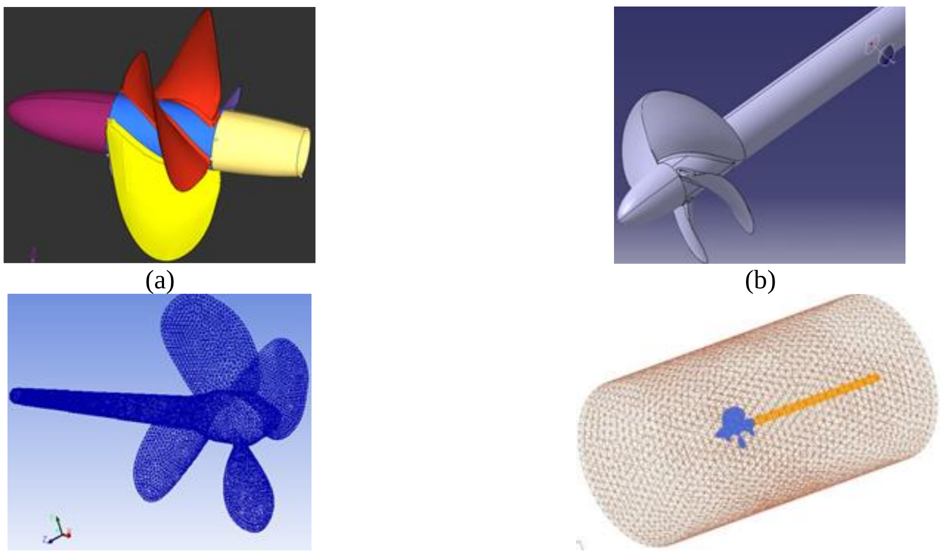
The Wageningen B-series being a general series expressed with open water diagrams obtained from open water or model tests where the K_T, K_Q, η_0, J curves are showed with constant blade number, blade area ratio and pitch ratios.

2.2. Flow simulation

The above governing equations have been converted to the algebraic equations which can be solved numerically using control volume technique of Fluent. This control volume technique consists of governing equations about each control volume yielding discrete equations and conserve the quantities on the basis of control volume. The standard wall functions used are based on the proposal of Launder and spangling and have been found quite reliable for propeller flows. The standard K-Ɛ model has been used.

3. NUMERICAL ILLUSTRATION

The computational domain was created for the scaled model propeller with a hub surrounded by a fluid domain shown in Figure 1(a, b). The inlet was prescribed at 3D upstream exit at 4D downstream solid surfaces on the blades and hub centred at the origin of the coordinate system followed by the alignment with uniform inflow. A hybrid mesh was generated inside the domain using hyper mesh 11.0 shown in Figure 1(d). Initially the blade surfaces were meshed with smaller triangular element while the inner region was filled approximately with growing triangular element and finally the domain region was filled with tetrahedral elements. The number of cells used in the mesh is about 583451. The 3D flow (viscous and incompressible) was solved using the FLUENT 6.3.26 code. The fluid continuum was chosen and assigned with the properties of water. A moving reference frame is assigned to fluid which has the same rotational velocity as that of propeller. The wall interface of the propeller blade and hub were assigned a relative rotational velocity of zero with respect to the adjacent cell zone. A uniform velocity was prescribed at inlet. The magnitude of velocity was adjusted to obtain appropriate advanced coefficients (J). At outlet outflow boundary condition was set. The far boundary (far-field) was taken as inviscid wall and assigned an absolute rotational velocity of zero. Fluent 6.3. Has been used for the computation of Flow through propeller (B-series). The simulation has been carried out to determine the propeller's open water characteristics with different varying advanced ratios in the range of 0.2- 1.4 as the requirement η_0 for the performance of the propeller in this operational range. The discretized fluid structure interaction can be seen in the Figure 2 where the propeller blade is surrounded by the tetrahedral element. The pressure distribution and rotational speed variation across the propeller blade can be observed in the Figure 3 and Figure 4 respectively. The comparison of presently Computed hydrodynamic characteristics responses of thrust coefficient (K_T), torque coefficient (K_Q) and open water efficiency (η_0) with that of the standard B-series data are presented in the Figure 5. It can be seen that the present hydrodynamic characteristics responses follow the same trend i.e. linear decreasing as in the standard B-series data after reaching to the 65 % of the advanced coefficient (J). The variation in the open water characteristic can be realised with respect to the advanced coefficient (J) in the Figure 6. The specification for the B-series propeller and the solver settings for the flow simulation are presented in the Table 1 and Table 2 respectively. Table 3 provide the more transparency in the numerical responses of the open water characteristics with the advanced coefficient (J) which was difficult to realise in the Figure 5. The variation in the Open Water Characteristics of B-Series Propeller with the varying rotational speed and the advanced velocity is shown in the Table 4 and Table 5 respectively. It is observed that all these responses show a significant decrement with a marginal change in the rotational speed and the advanced velocity.



(a) (b) (c) (d)

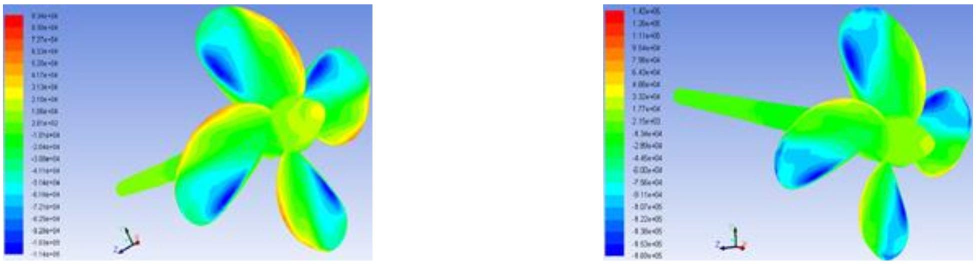
Figure 1. Different views and meshing of propeller with cap, hub, and shaft. (a) Side view of cap & hub (b) Propeller with cap and Shaft (c) Propeller meshed with tetrahedral elements (d) Propeller with control mesh

(a) (b)

Figure 2. Type of mesh surrounding propeller blade (a) Prism elements (b) Mesh connectivity

(a) Front side (b) Rear side

Figure 3. Pressure distribution around the propeller blade (a) Front side (b) Rear side



(a) N=1600rpm (b) N=2000rpm

Figure 4. Rotational speed variation across the blade & hub (a) N=1600rpm (b) N=2000rpm

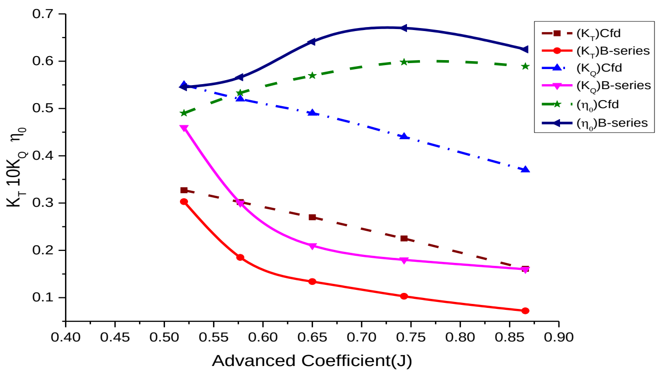


Figure 5. Comparison of predicted vs. B-series data

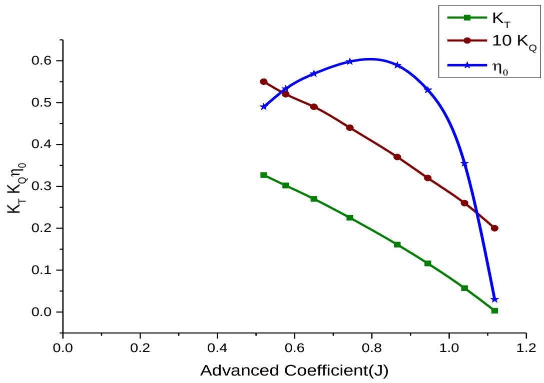


Figure 6. Open water characteristics for predicted-case-I

Table 1. Specifications of B-series propeller

Type of series	B-Wagenigen Series
Principal Dimensions	Propeller Diameter (D) = 0.353m
Domain size	Cylindrical domain of length 2.473m (7D), diameter 1.413m (4D)
Mesh count	~0.21 million tetrahedral and prism cells.
Number of blades (Z)	4
Rotational speed (N)	930rpm
Advanced velocity (Va)	6.12m/s

Table 2. Solver settings for flow simulation

Link of Pressure	SIMPLE
Pressure	Standard
Discretisation plan for convective fluxes with parameters of turbulence	upwind in Second order
model of turbulence	RNG (K-€)
Near Wall Treatment	Standard wall functions
Solver	Steady
Operating value of Pressure	120KPa

Table 3. Comparison of predicted and B-series data values of K_T, K_Q, η_o

Advanced coefficient (J)	Thrust Coefficient (K_T)		Torque Coefficient ($10K_Q$)		Efficiency (η_o)	
	Present	B-series data	Present	B-series data	Present	B-series data
0.866	0.161	0.072	0.037	0.016	0.589	0.625
0.743	0.225	0.103	0.044	0.018	0.598	0.670
0.650	0.270	0.134	0.049	0.021	0.5695	0.641
0.577	0.302	0.185	0.052	0.030	0.533	0.566
0.520	0.327	0.303	0.055	0.046	0.490	0.545

Table 4. Open water characteristics of B-series propeller (with varying speed and constant advanced velocity (V_a =6.12 ms-^{1})

Speed (N)	Thrust (T)	Torque (τ)	Advanced coefficient (J)	Thrust Coefficient (K_T)	Torque Coefficient (K_Q)	Open water efficiency (η_o)
33.33	5642.62	335.796	0.520	0.327	0.055	0.490
30.00	4225.31	258.930	0.577	0.302	0.052	0.533
26.66	2982.85	191.144	0.650	0.270	0.049	0.569
23.33	1906.76	132.569	0.743	0.225	0.044	0.598
20.00	1001.08	82.629	0.866	0.161	0.037	0.589
18.33	607.39	60.605	0.945	0.116	0.032	0.530
16.66	246.98	40.374	1.040	0.057	0.026	0.355
15.50	12.954	27.046	1.118	3.472e-3	0.020	0.030

Table 5. Open water characteristics of B-Series propeller with constant Speed n=15.5rps and varying advanced velocity

Advanced Velocity (V_a)	Thrust (T)	Torque (τ)	Advanced coefficient (J)	Thrust Coefficient (K_T)	Torque Coefficient (K_Q)	Open water efficiency (η_o)
2.0	1383.724	81.919	0.365	0.370	0.062	0.345
2.5	1299.282	76.616	0.456	0.348	0.058	0.434
3.0	1167.341	71.087	0.548	0.312	0.053	0.504
3.5	1019.356	65.342	0.639	0.273	0.0496	0.559
4.0	859.350	59.409	0.731	0.230	0.045	0.593
4.5	687.962	52.984	0.822	0.184	0.040	0.601
5.0	501.054	46.020	0.913	0.134	0.034	0.557
5.5	296.602	38.171	1.005	0.079	0.028	0.435
6.0	70.462	29.330	1.096	0.018	0.022	0.147
6.12	12.954	27.046	1.118	0.0037	0.020	0.030

4. CONCLUSIONS

The limiting streamlines and the path lines of the propeller blade, as well as the pressure distribution on the blade surface, represent its' flow characteristics more accurately. In absence of validation while employing the results if this CFD simulation solution generally tends to overestimate and the discrepancy increases with increasing propeller load i.e. decreasing (*J*). The deviation in solution may range from 5 to 15% in the case of thrust, 10-20% in case of torque. Pressure estimation is usually quite accurate with 5-8% variation. The tabular data of pressure distribution over the propeller blade may be used for further structural analysis. Maximum open water efficiency for computational result is 60.1% which is the instant exactly at the specific operating conditions which shows that the obtained computational results are up to the mark. In order to achieve a close form solution, tetrahedral element mesh has been replaced with hexahedral element mesh. There is a wide future scope of the present research work in finding more suitable method of combining blade and hub of a propeller within the block along with the refinement of mesh.

ACKNOWLEDGMENTS

Authors express their sincere regard to the National Institute of Technology, Rourkela-273010, Odisha and National Project Implementation Unit-Ministry of Human Resource and Development, Govt of India, for providing consistent support and encouragement in performing the present study.

NOMENCLATURE

N	Speed
V_a	Advanced velocity
J	Advancement Coefficient
K_T	Thrust Coefficient
K_Q	Torque Coefficient
η_0	Open water efficiency
n	Rotational speed in rps
D	Diameter of propeller
ρ	Density of material
$\frac{A_E}{A_0}$	Expansion ratio
$\frac{P}{D}$	Pitch to Diameter ratio
Z	Number of blades

REFERENCES

[1] Serkan EKINCI, "A Practical Approach for Design of Marine Propellers with Systematic Propeller Series," *Brodogradnja – Shipbuilding, ZNANOST / SCIENCE*, vol. 62, no. 2, pp. 123-129, 2011.

[2] Shiliang HU, ChuanjingLU and YoushengHE, "Fluid-structure interaction simulation of three-dimensional flexible hydrofoil in water tunnel," *Appl. Math. Mech. -Engl. Ed.*, vol. 37, no. 1, pp. 15–26, 2016.

[3] Burrill, L. C. and Yang, C. S, "The Effect of Radial Pitch Variation on the Performance of a Marine Propeller," *Transaction Institution of Naval Architects,* vol. 95, no. 5, pp. 119-132, 1953.

[4] Felli M., Camussi R., Di Felice F, "Mechanisms of evolution of the propeller wake in the transition and farelds," *Journal of Fluid Mechanics*, vol. 5, no. 2, pp. 682-695, 2011.

[5] Mihaela, A, "Developments in the design of ship propeller," *Transactions on Mechanics, University of Galati*, 2005.

[6] Miyata H, "Time-marching CFD simulation for moving boundary problems," *21st Symposium on Naval Hydrodynamics*, Trondheim, Norway, 1996.

[7] Subhas S., Saji V.F., Ramakrishna S., Das H.N, "CFD analysis of propeller flow and cavitation," *International Journal of Computer Application,* vol. 55, no. 4, pp. 26-33, 2012.

[8] Martinez, J., Balbona Calvo, L., Gonzalez Pérez, J., Blanco Marigorta E, "An Open Water Numerical Model for A Marine Propeller: A Comparison with Experimental Data," *Fluids Engineering Summer Conference,* Montreal, Canada, 2002.

[9] Brandt J.B., Selig M.S, "Propeller Performance Data at Low Reynolds Numbers," *49th AIAA Aerospace Sciences Meeting, AIAA*, 1211-1255, Orlando, FL, USA, 2011.

[10] Benini E, "Significance of blade element theory in performance prediction of marine propellers," *Ocean Engineering*, vol. 31, no. 9, pp. 957–974, 2004.

[11] Weick F.E, "Full Scale Wind Tunnel Tests with a Series of Propellers of Different Diameters on a Single Fuselage," *NACA Rep*. vol. 8, no. 4, pp. 339-351, 1930.
[12] Weick F.E, "Full Scale Wind Tunnel Tests on Several Metal Propellers Having Different Blade Forms," *NACA Rep*., vol. 13, no. 7, pp. 339-351, 1930.
[13] Justin, E, "Marine propellers," *Annual Review of Fluid Mechanics*, vol. 18, no. 6, pp. 367-403, 1986.
[14] Valentine D.T, "Reynolds-averaged Navier-stokes codes and marine propulsor analysis," *Hydrodynamics Directorate Research and Development Report,* Carderock Division, Naval Surface Warface Centre, Bethesda, MD., USA 1993.
[15] Kerwin J.E. and Lee C.S, "Prediction of Steady and Unsteady Marine Propeller Performance by Numerical Lifting-Surface Theory," *Transaction SNAME*, vol. 86, no. 4, pp. 218–253, 1978.
[16] Streckwall H, "A Method to Predict the Extent of Cavitation on Marine Propellers by Lifting-surface-theory," *International Symposium on Cavitation, Sendai, Japan*, 1986.
[17] Kim H.T. and Stern F, "Viscous flow around a propeller shaft configuration with infinite-pitch rectangular blades," *Journal of Propulsion*, vol. 13, no. 6, pp. 434–443, 1990.
[18] Ghassemi H. and Ghadimi P, "Computational hydrodynamic analysis of the propeller–rudder and the AZIPOD systems," *Ocean Engineering,* vol. 34, no. 6, pp. 117–130, 2007.
[19] Ghassemi H, "The effect of wake flow and skew angle on the ship propeller performance," *Scientia Iranica*, vol. 16, no. 2, pp. 149–158, 2009.
[20] Ghassemi H, "Hydrodynamic performance of coaxial contra-rotating propeller [CCRP] for large ships," *Polish Maritime Research,* vol. 16, no. 59, pp. 22–28, 2009.
[21] Ghassemi H, "Hydrodynamic characteristics of the surface-piercing propellers for the planning craft," *Journal of Marine Science and Application,* vol. 11, no. 7, pp. 147–156, 2008.
[22] Ghassemi H, Ghassabzadeh M, and Saryazdi MG, "Effect of material on hydro-elastic behaviour of marine propeller by using BEM-FEM hybrid software," *Polish Maritime Research,* vol. 20, no. 9, pp. 62-70, 2013.
[23] Young Y.L. Fluid-structure interaction analysis of flexible composite marine propellers," *Journal of Fluids and Structures*, vol. 24, no. 13, pp. 799-818, 2008,
[24] Young, Y. L. And Savander BR, "Numerical analysis of large-scale surface-piercing propellers," *Ocean Engineering*, vol. 38, no. 13, pp. 1368–1381, 2011.
[25] Dymarski P, "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, no. 11, 2008.
[26] Prakash S. and Subramanian VA, "Simulation of propeller- hull interaction using RANSE solver," *International Journal of Ocean and Climate Systems*, vol. 1, no. 3, pp. 189-203, 2009.
[27] Das H.N, "CFD Analysis for cavitation of a Marine Propeller," *8th symposium on High speed Marine Vehicles*, Naples, Italy, 2008.
[28] Das H.N., Jayakumar L.C.P, "Computational prediction and Experimental validation of the Characteristics of a contra rotating propeller," *NRB seminar on Marine Hydrodynamics*, Visakhapatnam, India, 2002.
[29] Banerjee CN, Das HN and Srisudha B, "Computational analysis and experimental validation of hull propulsor interaction for an autonomous underwater vehicle (AUV)," 7th *Asian CFD Conference*, Bangalore, India, 2007.
[30] Krishna GV, Saji VF, Das HN, and Panigrahi PK, "Acoustic Characterization of a Benchmark Marine propeller using CFD," *Naval Symposium on Acoustics (NSA),* Visakhapatnam, India, 2008.
[31] Krasinikov V, Zhang Z and Hong F. Analysis of Unsteady Propeller Blade Forces by RANS," 1st *International symposium on Marine Propulsors*, Trondheim, Norway, 2009.
[32] Subramanian VA and Senthil PMN, "Body force-based simulation of propeller hull interaction," *3rd International Conference in Ocean\ Engineering*, IIT Madras, Chennai, India, 2009.
[33] Kimura K, Kawamura T, Huang Z, Fuji A and Taketani T, "Study on unsteady cavitating flow simulation around marine propeller using a RANS CFD code," *7th International Symposium on Cavitation*, Ann Arbor, MI, 2009.
[34] Takashi. K., Jun A, "Numerical analysis of steady and unsteady sheet cavitation on a marine propeller using a simple surface panel method SQCM," *1st International symposium on Marine propulsions*, Trondheim, Norway, 2009.
[35] Choong W.H., Yeo K.B., Tamiri F. and Teo K.T.K, "Outboard marine propeller performance analysis through CFD modeling," *15th International Conference on Computer Modelling and Simulation*, Cambridge, U.K. 2013
[36] Madaci M. and Kerdoun D, "A CFD Study with Fuzzy Logic Pitch Angle Turbine Control Implementation for OWC Generation System," *International Journal of Advances in Applied Sciences (IJAAS)*, vol. 6, no. 2, pp. 175-184, 2017.
[37] Das H.N. and Kapuria S, "On the use of bend–twist coupling in full-scale composite marine propellers for improving hydrodynamic performance," *Journal of Fluids and Structures*, vol. 11, no. 6, pp. 1132–153, 2011.