

Open Water Performance of a Marine Propeller Model Using CFD

M. A. Elghorab, A. Abou El-Azm Aly, A. S. Elwetedy, and M. A. Kotb

Abstract— In general, marine propeller blades have complicated geometries and as a consequence, the flow around these propellers is complicated. Propeller tests in open water are commonly used to obtain the graphs of the hydrodynamic performance.

In the present paper, a numerical simulation of a marine propeller model using a commercial code, ANSYS CFX, is used to figure out the open water performance of this propeller for different operating conditions. The effects of mesh density and type of turbulent model on performance are investigated.

The obtained results have been directly compared to experimental results for the same marine propeller model and for the same operating conditions previously published to justify the CFD results.

The numerical investigation has been extended to increase the studied range. In addition, a direct plot of B-series with this validated results have been plotted to show the propeller trend.

Keywords— Marine propeller, CFD, open water performance.

I. INTRODUCTION

Marine propellers are designed to provide the maximum thrust and the minimum torque for the optimum propeller rotation speed and the required cruising ship speed which allows ships to efficiently travel at their design speed. However, it is important to know how the marine propeller performs during off-design situations as well as design point by knowing its open water performance. This is important for all naval marine propellers whose missions require them to perform at a wide range of speeds.

Choosing the correct marine propeller is a vital design step to achieve the required performance, as the marine propeller is the principal ship component to provide the necessary propulsion. In real situations, assessment of marine propeller behind hull is still difficult since a highly disturbed wake field behind the hull exists. Consequently, a widely known approach is the open water performance one to assess the marine propeller. In open water test, in which the propeller to be tested without the hull, the performance characteristics relate thrust and torque coefficients with the flow advance speed ratio.

The efficiency of the propulsion system is strongly dependent on propeller performance, thrust force, torque of propeller and its efficiency. Therefore, the simple method for investigation of marine propeller hydrodynamic performance is used to getting graphs of the propeller performance coefficient (K_T , K_q , η) with respect to advance coefficient (J).

Martinez-Callewas *et al* [1] tested fishing-boat propeller numerically using a three-dimensional unstructured mesh. Mesh dependency and different turbulent models are considered together with a sliding technique to account for the rotation. Typical turbomachinery boundary conditions for a volume containing the propeller are imposed (inlet velocity and outlet static pressure).

While Nakisa *et al* [2] approached the propeller hydrodynamic performance via numerical modeling using a finite volume commercial code such as Fluent. Modeling of the propeller was based on RANS equations in steady flow with unstructured mesh. The effect of mesh density, type of turbulent model and numerical solution algorithm on modeling were investigated. Finally, the results of the simulations were compared with available experimental data for the selection of the best modeling methodology for open water tests of the propeller.

While Krasilnikov *et al* [3] investigated the scale effect on characteristics of marine propellers using a RANS method, and focused on the aspects related to the influence of blade skew, propeller loading, and blade area ratio. The flow was assumed to be fully turbulent and the blade surface was smooth. The differences in forces and relevant scale effects on propellers with different magnitude of skew were demonstrated and explained through the analysis of flow patterns around the blades and through the estimation of pressure and friction force contributions.

While Amoraritei [4] investigated the hydrodynamics performances and flow fields around conventional propellers and azimuth thrusters numerically. The propeller running at azimuth thruster was influenced by many interactions between propeller and the gondola and strut. The geometry of propeller, gondola and strut were known and the problem was solved using a 3D model and a commercial code FLUENT.

In the present paper, the open water marine propeller performance has been conducted numerically to illustrate the thrust and torque coefficients of a marine propeller model using commercial code, ANSYS CFX, the results have been validated with experimental one of the same propeller [5].

A direct comparison between the obtained numerical results and a theoretical study of a B-series marine propeller of the same blade area ratio has been carried out.

M. A. Elghorab is lecturer Assistant, Department of Mechanical Power and Energy, Military Technical College, Cairo, Egypt. (Corresponding Author email: m_elghorab43@hotmail.com).

A. Abou El-Azm Aly and A. S. Elwetedy are lecturers at the Department of Mechanical Power and Energy, Military Technical College, Cairo, Egypt.

M. A. Kotb is prof. in Marine Engineering and Naval Architecture Department, Alexandria University, Alexandria, Egypt.

II. NOMENCLATURE

D	Propeller diameter, [m].
J	Advance coefficient, [-]
K_Q	Torque coefficient, [-].
K_T	Thrust coefficient, [-].
k	Turbulent kinetic energy, [m^2/s^2].
n	Rotational speed, [rps].
P	Propeller pitch, [m].
Q	Torque at the propeller axis, [$\text{N}\cdot\text{m}$].
T	Thrust of the propeller, [N].
V_a	Advance velocity, [m/s].
x	Axial coordinate, [m].
ε	Turbulent dissipation, [m^2/s^3].
ρ	Density of the water, [Kg/m^3].
η	Propeller efficiency, [-].

III. PROPELLER DESCRIPTION

The geometry considered during this study is a marine propeller model, that is, three blade clock-wise (right-hand) with an outlet diameter of $D = 198$ mm and a pitch to diameter ratio of $P/D = 1$ at $r/R=0.7$, as shown in Figure (1).



Fig. (1) Commercial marine propeller under investigation

To achieve this study, the propeller, to be analyzed, has been digitized using coordinate measuring machine to produce the blade section data then NX 7.5 software which has been able to loft and merge blade sections to produce the whole blade shape with refining as shown in Figure (2), then it has been imported to the ANSYS CFX as a solid part.

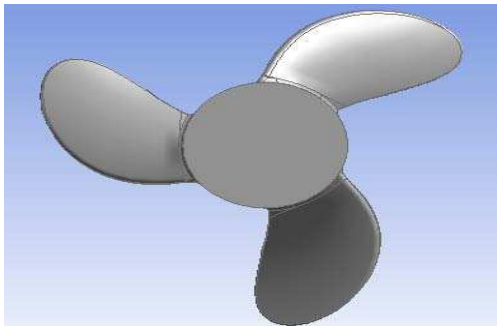


Fig. (2) Imported propeller to CFX

IV. COMPUTATIONAL DOMAIN

In studying the propeller computational domain, two domains have been created as shown in Figure (3), one

domain was for the propeller enclosure which is a rotating domain. The other domain represents the fluid around the propeller domain which is a stationary domain. The rotating domain rotates around x-axis with a specific angular speed (rpm) which is different from case to another. The working fluid is water at normal temperature and pressure.

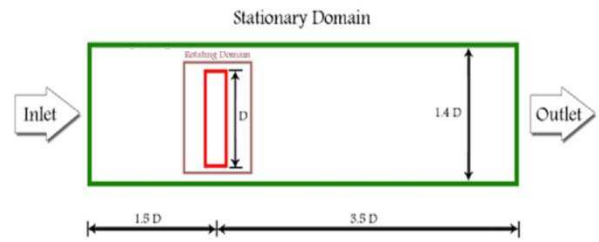


Fig. (3) Propeller computational domain

The rotating domain which encloses the propeller has the following dimensions: radius of 6 mm, length in positive x direction of 14 mm and length in negative x direction of 14 mm. While stationary domain has the cylinder shape with the following dimensions [6], the stationary fluid upstream domain has a length of $1.5 D$, 297 mm, from the propeller domain while the downstream domain has $3.5 D$, 693 mm, in length. The stationary fluid domain has a diameter of $1.4 D$, 277.2 mm as shown in Figure (4). (all the previous dimensions are measured from the propeller border).

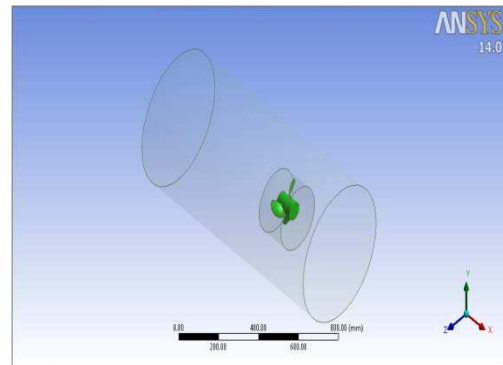


Fig. (4) Propeller domains in ANSYS

V. GRID CREATION

Due to the complexities in the marine propeller geometry, unstructured mesh has been selected to adapt this complicated geometry. Different zones with refined meshes have been introduced around the cross section of the blade at the hub intersection and near blade tip where high velocity gradients have been expected to enhance the accuracy of the solution.

Different grid sizes for the studied 3-D propeller geometry have been investigated to check the solution dependency on this grid size which have been named as medium, medium +50, fine and fine +50 (+50 means here that the medium or fine mesh have been refined by 50%).

In applying the coarse mesh in this study, the mesh elements were large in size with respect to the angles at the hub blade intersection, so it has been ignored in this test. This comparison between the different sizes of mesh elements have been done at specific conditions of propeller rotational speed of 445 rpm and advanced velocity of 0.357 m/s, and the resultant thrust and torque have been calculated

and thrust and torque coefficients have been tabulated with that of experimental results as shown in Table (I).

Experimental values for thrust and torque coefficients for this validated case are K_T is 0.1845 and K_Q is 0.0203 [5].

Table (I) Grid Sensitivity Analysis

Total number of elements	Medium	Medium+50	Fine	Fine+50
614035	1213418	1794647	3635466	
T (N)	17.46	17.31	15.82	15.74
KT	0.21	0.2	0.19	0.19
Error %	11.98	11	1.4	0.93
Q (N.m)	0.55	0.49	0.37	0.39
KQ	0.03	0.03	0.02	0.02
Error %	62.06	43.34	9.5	15.27
Physical time	2h 6min 24s	5h 4min 24s	11h 13min 19s	19h 4min 21s

It has been shown that the error between the CFD results and that of the experimental for thrust and torque coefficients have been decreased as the mesh element has been refined. As a result, the fine mesh has been selected to be used in the current study because the error between the CFD results and that for experimental is not far from that of fine +50. In addition, the computational time has been reduced, a detailed mesh for the studied domain has been shown in Figure (5).

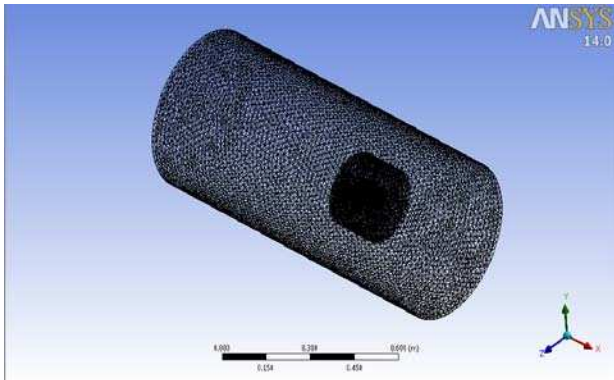


Fig. (5) Fine mesh of the propeller computational domain

VI. TURBULENCE MODELING

These approaches to numerical solution of the governing equations introduce unknown variable and turbulence models are needed to describe the effects of turbulence and to determine the “additional terms”. It is known that no single turbulence model is universally accepted as being superior for all kinds of turbulent flows.

The accuracy required by the particular applications and the computational efforts must be taken into account at turbulence model selection. “In many cases a numerically inexpressive turbulence model can predict some global measures with the same accuracy as a more complex model”[4].

In this study a comparison between the three studied models $k-\epsilon$, $k-\omega$ and the $k-\omega$ based SST model have been conducted with the same setup and geometry to choose the most suitable model to solve the studied case and the results have been tabulated as shown in Table (II):

Table (II) Turbulence Model Selection

Model	Standard $k-\epsilon$	Standard $k-\omega$	Standard SST
T (N)	17.27	17.27	17.35
K_T	0.2	0.2	0.21
Error %	10.67	10.67	11.21
Q (N.m)	0.39	0.38	0.37
K_Q	0.02	0.02	0.02
Error %	15	13.26	9.5
Physical time	3h 7min 11s	2h 52min 40s	2h 59min 49s

For the studied case, the experimental results are K_T is 0.1845 and K_Q is 0.0203. The SST model has been developed to overcome deficiencies in the $k-\omega$ and BSL $k-\omega$ models and Based on the comparison in Table (II), The SST has been recommended to study the propeller analysis in open water performance, so it has been selected in this study.

VII. BOUNDARY CONDITIONS

Boundary conditions have been specified the flow variables on the boundaries of the physical model. There are two portal Boundary conditions found in the computational domain the inlet flow port and the outlet flow port as shown in Figure (6).

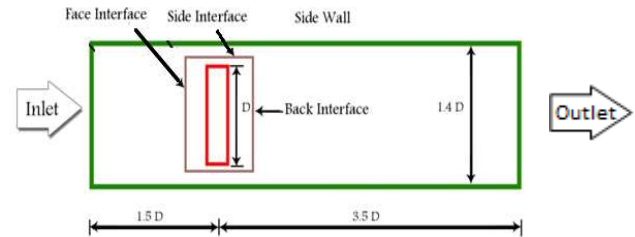


Fig. (6) Boundary conditions schematic

The inlet boundary condition has been defined by the axial flow velocity normal to the boundary and the outflow boundary condition has been used to model flow exits from the stationary domain by averaging the static pressure at the outlet. Stationary domain side has been considered to be as a smooth wall, there is no flow in and out from that side. In this study the type of the interface is fluid to fluid interface.

VIII. NUMERICAL PROCEDURES

The continuous information has been contained in the exact solution of the differential equation describing the physical model would have been replaced with discrete values, in which, all grid nodes would have interactive properties between them and the neighbor nodes. A finite volume technique has been used in CFX package to convert the governing equations to algebraic equations that can be solved numerically.

The convergence can be monitored dynamically by checking residuals. The residuals must be kept on decreasing from the start to end of the iterations, in this study the scaled residuals decrease to 10^{-5} for all equations. The numerical results have been obtained using about 5032 iterations to obtain a suitable level of solution convergence

as shown in Figure (7) which shows the residuals history versus the number of iterations.

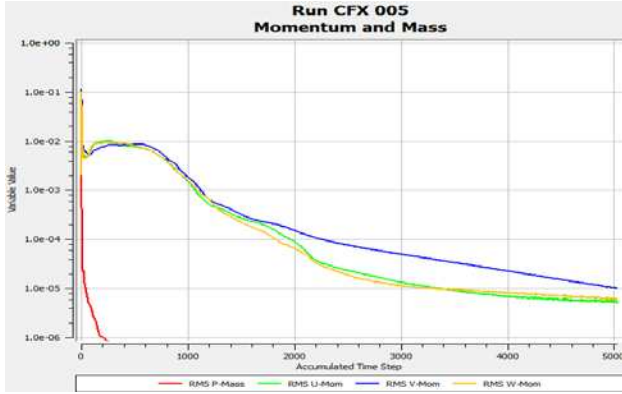


Fig. (7) Residuals history with the number of iterations

IX. NUMERICAL SIMULATION VALIDATION

A direct comparison between numerical results and the experimental results has been carried out to validate the numerical model in the studied range. Validation has been carried out by calculating the thrust and torque coefficients at cases prescribed in experimental work results. The efficiency has been calculated and has been plotted to complete the total behaviour of the marine propeller model.

Six different cases have been investigated as shown in Table (III):

Table (III) Different studied cases

Case	N (rpm)	V_A (m/s)	J
Case A	585	0.357	0.185
Case B	645	0.357	0.168
Case C	512	0.35	0.207
Case D	667	0.35	0.159
Case E	450	0.179	0.121
Case F	550	0.179	0.09

Figure (8) shows the thrust coefficient which has been resulted from the numerical simulation for the tested propeller model. The numerical simulation results have been in a good agreement with those obtained from experimental test. The difference between the numerical simulation results and the experimental results is 10.59 % which may be due to some errors in the experimental measurements.

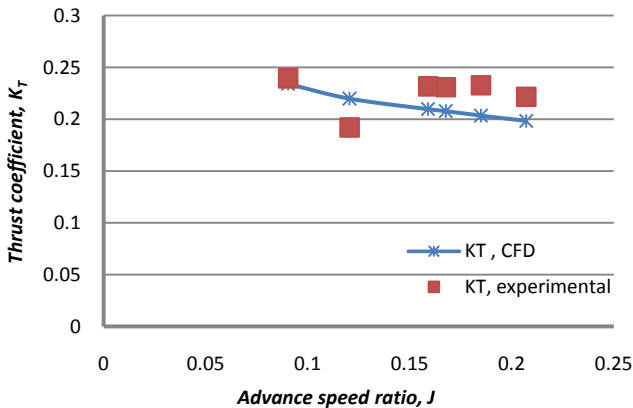


Fig. (8) Numerical versus experimental thrust coefficient

Figure (9) shows the numerically obtained torque coefficient compared with the experimental results. The numerical simulation results have been in a good agreement in behaviour with that obtained from these experimental results. The error between the numerical simulation results and the experimental results is 18.9 % which may be due to some errors in the experimental measurements.

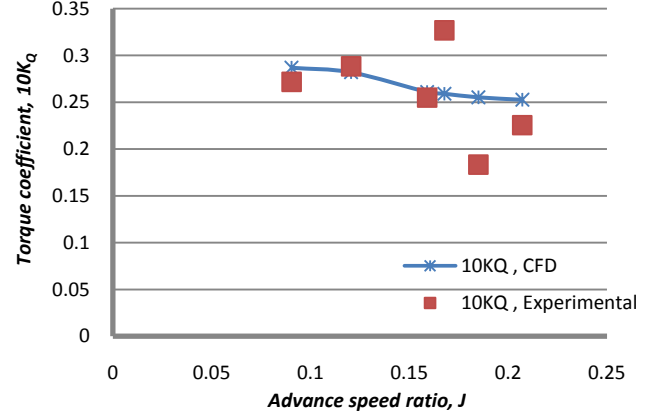


Fig. (9) Numerical versus experimental torque coefficient

Figure (10) shows the overall open water performance of the tested marine propeller resulted from numerical simulation.

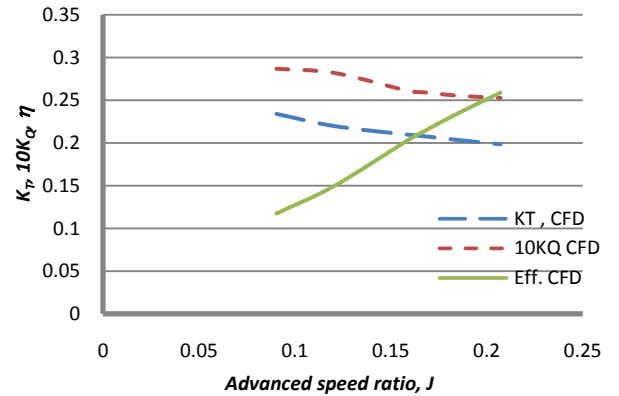


Fig. (10) Overall numerical performance

X. CFD RESULTS

Six different cases have been investigated and analysed from which the pressure distributions and velocity profiles on the propeller blade have been obtained as follows:

Figures (11) through (16) show the total pressure distribution on the pressure side of the blade for cases A, B, C, D, E and F. These figures illustrate that how the pressure on the blade face increases with increasing of rpm at constant advanced velocity. The highest value of the pressure on the blade at blade leading edge as it is the first point meets water.

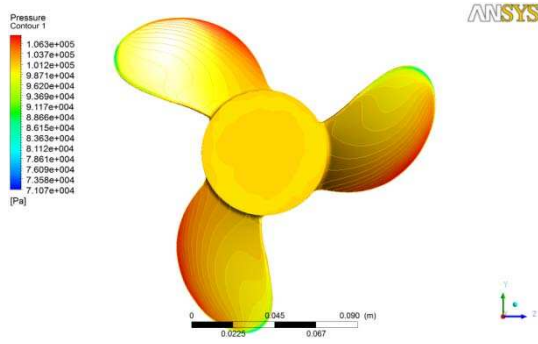


Fig. (11) Pressure distribution on propeller face, Case A

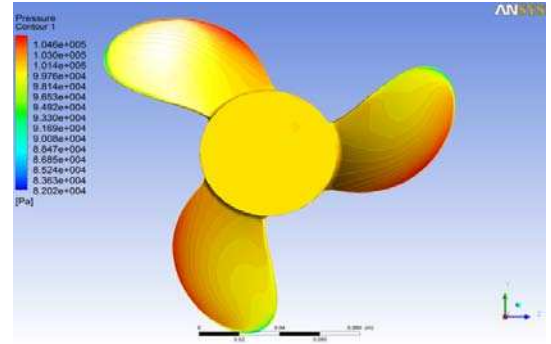


Fig. (15) Pressure distribution on propeller face, Case E

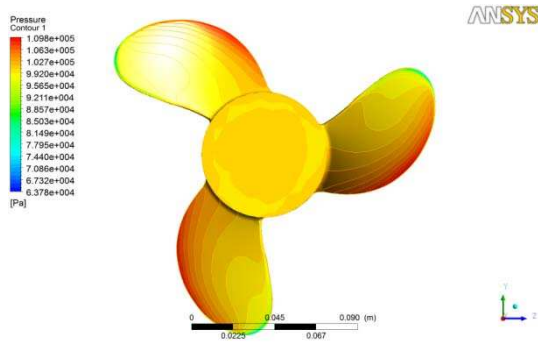


Fig. (12) Pressure distribution on propeller face, Case B

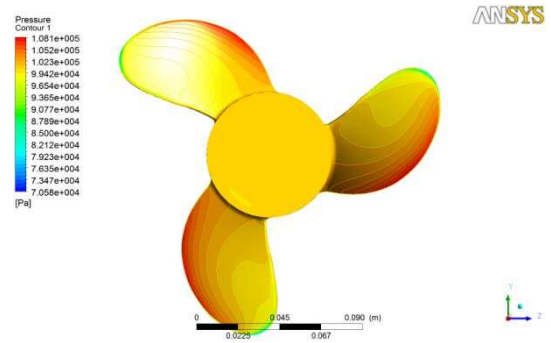


Fig. (16) Pressure distribution on propeller face, Case F

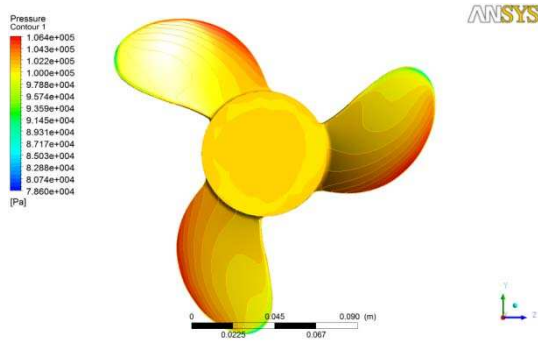


Fig. (13) Pressure distribution on propeller face, Case C

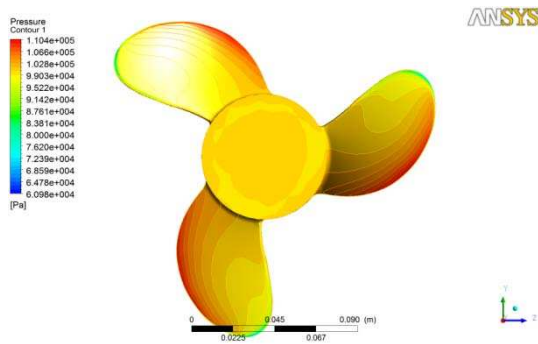


Fig. (14) Pressure distribution on propeller face, Case D

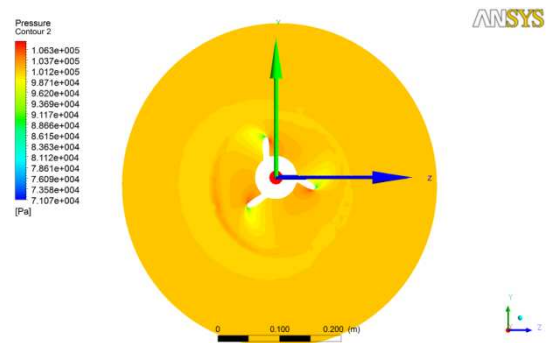


Fig. (17) Pressure distribution at propeller plane, Case A

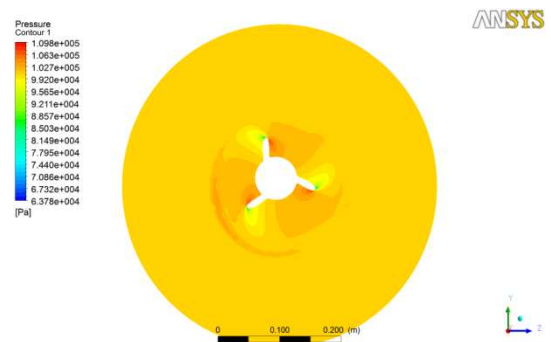


Fig. (18) Pressure distribution at propeller plane, Case B

Figures (17) through (22) show pressure distribution on blade face at propeller section. As seen pressure increasing at the face of the blade, pressure face, with increasing rpm especially at the blade tip. Also there is pressure decrease at the back of the blade.

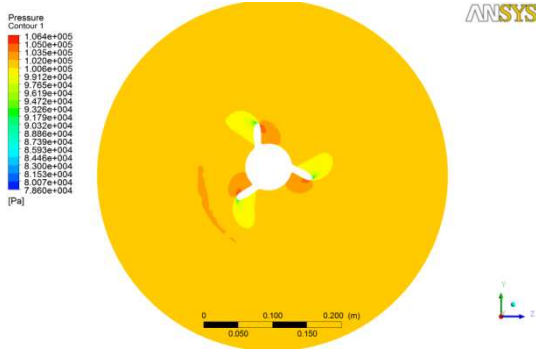


Fig. (19) Pressure distribution at propeller plane, Case C

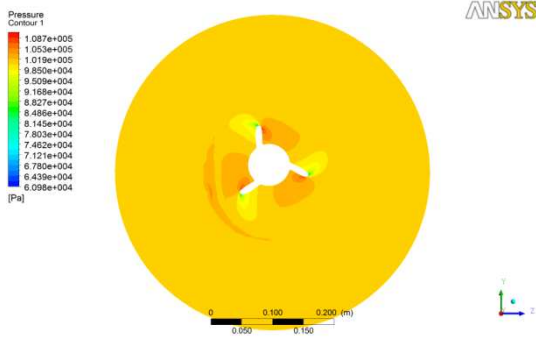


Fig. (20) Pressure distribution at propeller plane, Case D

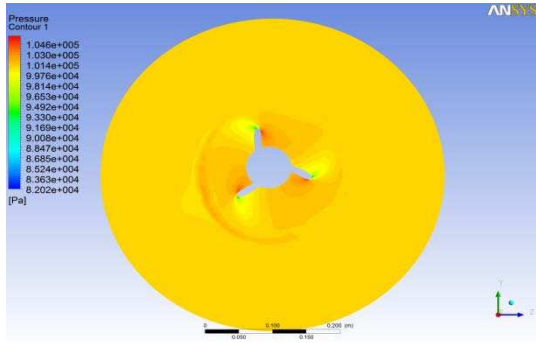


Fig. (21) Pressure distribution at propeller plane, Case E

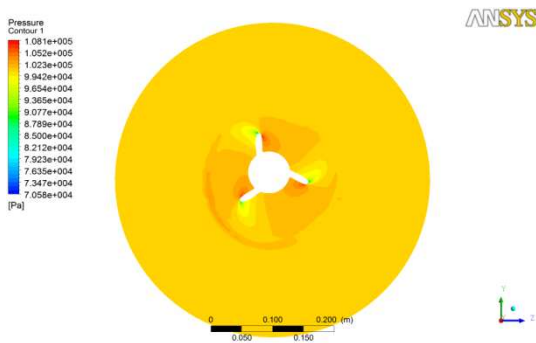


Fig. (22) Pressure distribution at propeller plane, Case F
at propeller boundary, Case A

The pressure and axial velocity distributions with the propeller radius at propeller plane have been shown in Figures (23) and (24).

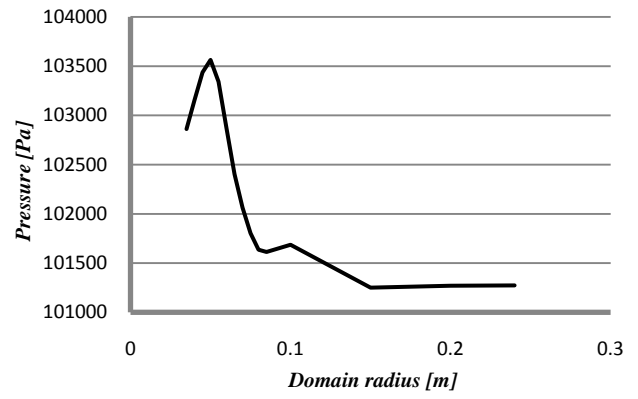


Fig. (23) Pressure distribution with the propeller radius at propeller plane.

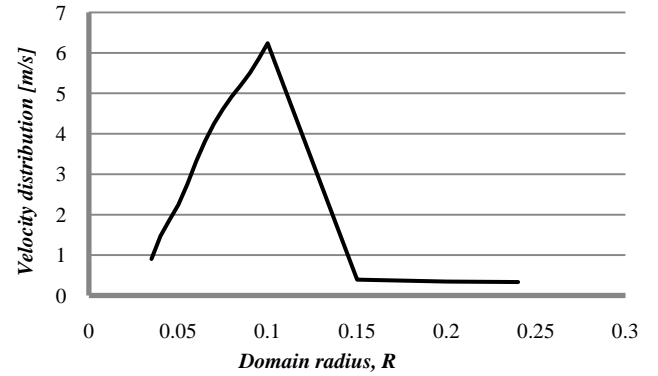


Fig. (24) Axial Velocity profile with the propeller radius at propeller plane.

XI. EXTENDED CFD RESULTS

After the validation of CFD results with that of experimental results in the range of advanced speed of 0.0904 to 0.2431, this limited range due to the limitation of the experimental test facilities [5], the CFD simulation has been extended to cover the range of advanced speed ratio up to 0.8 as shown in Figure (25) the extended CFD result have been compared with a B-series result of the same propeller geometry to show the overall studied propeller trend as shown in Figure (26).

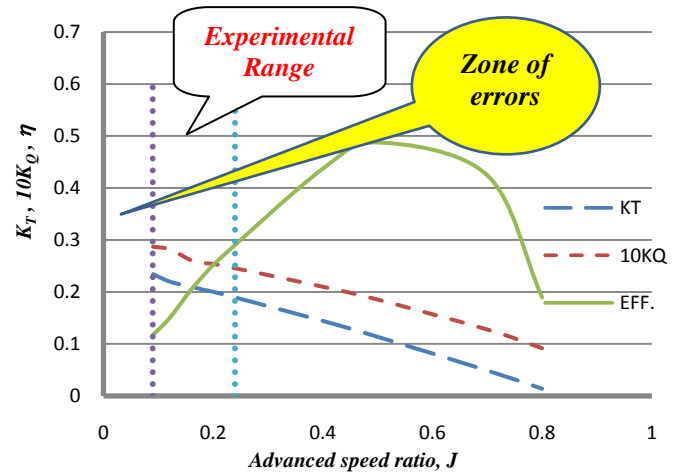


Fig. (25) Extended CFD results

The extended CFD result have not been tested in the range of advanced speed ratio from zero to 0.09 as this region is a zone of errors because the lift theory is applied to get the performance in this region while in the rest region the momentum theory has been applied [7].

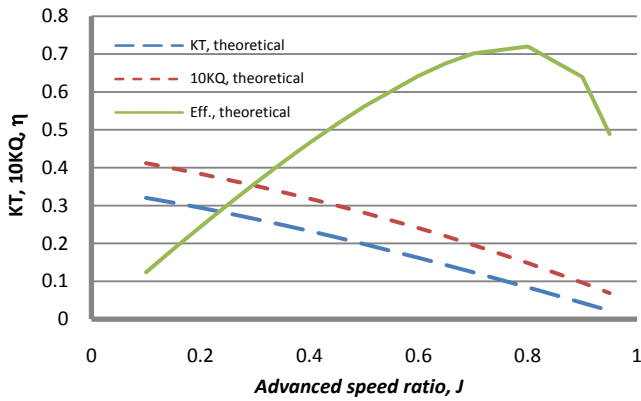


Fig. (26) B-series propeller performance

XII. CONCLUSION

A numerical simulation has been carried out to obtain an marine propeller model performance using ANSYS CFX package and the results have been concluded that numerical simulation of the marine propeller model performance has been carried out using a commercial code, ANSYS CFX. It is important to choose the suitable mesh model of the case study to get the optimum efforts and time. A direct comparison of the numerical results with an experimental one of the same propeller parameters has been carried which explain that the behaviour of the tested propeller is similar to the behaviour of B-series marine propeller, which has the same BAR, D, P/D and no. of blades, propeller but with different scale. Finally, The numerical results are in a good agreement with experimental results with an error of 10.59 % for thrust coefficient and 18.9 % for torque coefficient. The extension of the CFD simulation has been extended to have a range of 0.0904 to 0.9 and has been proved that its trend is in a good agreement with that of B-series curves.

REFERENCES

- [1] J. Martinez-Calle, L. Balbona-Calvo, J. Gonzalez-Pérez, E. Blanco-Marigorta, "An Open Water Numerical Model for A Marine Propeller: A Comparison with Experimental Data" Proceedings of the ASME FEDSM'02 2002 Joint US-European Fluids Engineering Summer Conference, Montreal, Canada, FEDSM2002-31187, July 2002.
- [2] M. Nakisa, M. J. Abbasi, A. M. Amini, "Assessment of Marine Propeller Hydrodynamics Performance in Open Water Via CFD," Vol. Proceedings of Martec 2010, The International Conference on Marine Technology, BUET, Dhaka, Bangladesh, 2010.
- [3] V. Krasilnikov, J. Sun, K. H. Halse, "CFD Investigation in Scale Effect on Propellers With Different Magnitude of Skew in Turbulent Flow," in The First International Symposium on Marine Propulsors, Trondheim, pp. 25-40, 2009.
- [4] M. Amaraitei, "Application of CFD in Analysis of Conventional Propeller and Azimuth Thruster," Scientific Bulletin of the 'Politehnica' University of Timisoara, Transactions on Mechanics, Vol. 50, 2005.
- [5] M. A. Elghorab, A. Abou El-Azm Aly, A. S. Elwetedy, and M. A. Kotb "Experimental Study of Open Water Non-Series Marine Propeller Performance", Proceeding of International Conference on Fluid Mechanics, Heat Transfer and Thermodynamics, ICFMHTT, Copenhagen, Denmark, June 2013.
- [6] J. Kulczyk, L. Skraburski, M. Zawislak, "Analysis of Screw Propeller 4119 Using the Fluent System," Archives of Civil and Mechanical Engineering, Vol. 7, pp. 129-137, 2007.
- [7] E. Benini, "Significance of Blade Element Theory in Performance Prediction of Marine Propellers," Ocean engineering, , Vol. 31, pp. 957-974, 2004.
- [8] ANSYS 14.5.7.