Proceedings of MARTEC 2010
The International Conference on Marine Technology
11-12 December 2010, BUET, Dhaka, Bangladesh

ASSESSMENT OF MARINE PROPELLER HYDRODYNAMIC PERFORMANCE IN OPEN WATER VIA CFD

Mehdi Nakisa[1], Mohammad Javad Abbasi[2], and Ahmad Mobasher Amini[3]

[1]Islamic Azad University
Boushehr Branch, Iran
mnakisa2003@yahoo.com

[2]IGS
Mood582003@yahoo.com

[3]Assistan Professor
Persian Gulf University, Iran
mobasheramini@pgu.ac.ir

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. The aim of this paper is to approach the propeller hydrodynamic performance via numerical modeling using a finite volume commercial code such as Fluent. Modeling of a general B-Series propeller is 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 are investigated. In order to get the open water test performance coefficients for the considered propeller (KT, KQ, η) different advance coefficients (J) are imposed as boundary condition for the numerical model. Finally, the results of the simulations are compared with available experimental data for the selection of the best modeling methodology for open water tests of the propeller.

***Key words**: RANS, CFD, marine propeller performance*

NOMENCLATURE

D: Diameter of Propeller, [m]
Z: Number of blades
$\frac{A_E}{A_0}$ = Expanded area ratio
$\frac{P}{D}$ = Pitch ratio
K_t: Thrust coefficient, $K_T = \frac{T}{\rho n^2 D^4}$
K_q: Torque coefficient, $K_Q = \frac{Q}{\rho n^2 D^5}$
q: Torque at the propeller axis, [n.m]
Br: Boss ratio (br=b/d)
r: Rake angle, [deg.]
V_a: Advance velocity, [m/s]
n: Rotational speed, [rpm]
ρ : Density of the water,[kg/m2]
t: Thrust of the propeller
J: Advance coefficient, J=$\frac{V_A}{nD}$
η : Propeller efficiency, $\eta = \frac{K_T}{K_Q}\frac{J}{2\pi}$

1. INTRODUCTION

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 assessment of marine propeller hydrodynamic performance is used to graphs of the propeller performance coefficient (KT, Kq, η) with respect to advance coefficient (J). Different numerical methods for the treatment of the RANS (Reynolds Average Navier Stokes) equations have been widely developed.

Although predictions on the flow field are always valuable, the numerical simulation of marine propellers is difficult. These difficulties lie in the usual CFD difficulties such as: turbulence modeling, flow separation, boundary layer etc. Kodama (1992) and Funeno (1997) used a RANS equation method with structural mesh technique for simulation of the current around a ship's hull and propeller [3]. This method has some difficulties such as: restriction of results only in steady state and open water, also because of blade's geometry complication. They couldn't predict the current near the tip of blades accurately. Funeno (1999 and 2002) simulated the current around a highly skewed propeller via unstructured mesh [4]. The results had good correspondence experiment data for steady state and unsteady flow, but this method was complicated and time consuming. Also Martinez (2002) simulated the propeller via k-epsilon turbulent method in open water in steady state condition. Totally, the results were acceptable but there was approximately 30% error in prediction of torque coefficient [6]. Takekoshi (2003) simulated the propeller via standard k-omega turbulent method in open water and steady state conditions. For simulation of propeller geometry and its surrounding, he used the propeller symmetry and simulated only one blade and compared the results of experiments with his simulated result; 15 percent error in the simulation results. [7]

Rhee simulated 2D propeller model in unsteady state flow. He used the sliding mesh method for displaying the propeller rotation and turbulent flow of standard k-omega. Its error in comparison with experimental method was approximately 13 percent [8].

2. MODEL AND EXPERIMENTAL MEASUREMENTS:

B-Wageningen series is one of the well-known standard propeller series which is used in propulsion system of merchant ships.

The range of P/D in these series of propellers is 0.6 to 1.4 with Four to Seven blades.

Four blades one are used commonly for merchant ships. Therefore in our study a four blades B-Wageningen standard propeller simulated with the following characteristics

model name: B-Wageningen

Z: 4

D: 0.275

Br: 0.1670

$\frac{P}{D}$ at 0.7R: 0.7

$\frac{A_E}{A_0}$: 0.4

R: 10 deg.

We have used the experimental results of hydrodynamic coefficient performance in open water test [5].

- Stationary zone: this zone is cylindrical and includes the boss, propeller and moving zone. According to Takekoshi [1], upstream length is taken as 2D, downstream length is 5D and zone diameter is 3D, as shown in Fig.1. where D is the diameter of the propeller.

2.1 BOUNDARY CONDITIONS AND NUMERICAL SOLUTION CONTROL:

For simulation the propeller in steady state flow, we divided the calculation zone in two cylindrical divisions:

- Moving zone: length and diameter of this zone are depended on diameter and boss of the propeller. The aim of moving zone is simulating the movement of propeller and boss rotation and applying the Coriolis acceleration term in the Navier Stokes equations.

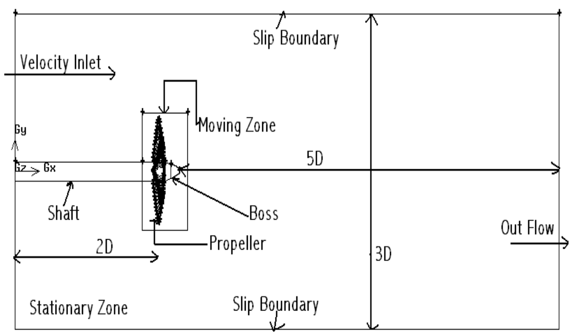


Fig.1: The scheme of propeller, Boss and numerical domain

The water in our simulation is incompressible flow; two different approaches have been tested, for the inlet boundary condition:

- Inlet flow velocity
- Inlet total pressure

The inlet flow velocity condition is selected, because it is considered with more physical meaning.

For modeling the inlet flow velocity, we used the stationary zone and velocity of flow.

We used two options for simulation of the outlet boundary condition:

- Outflow
- Pressure outlet

Though outlet flow condition, increases calculation time and decreases convergence rate. We used it for modeling of stationary zone, since the outlet flow condition has more physical meaning than outlet pressure condition. For simulation the boundary condition around stationary zone, has been used the slip boundary condition, in order to decrease the wall influence on stream flow as shown in Fig.1.

2.2 MESHING AND CALCULATION ZONE:

A geometrical discretization of the propeller is made for the treatment. Unstructured tetrahedral cells are used to define a control volume (symmetric with respect to the propeller axis).

A mesh refinement zone is defined near the propeller surface in order to capture the phenomena better at the propeller blades.

We must exert the momentum, conservation of mass and energy equation on flow field. We can not say that small mesh is good mesh because we must consider the time and cost calculation.

Final model has around 1,100,000 cells. Grids on the propeller surface are triangle shape that of various sizes because the cells near the root, blade edges and tip of propeller are smaller than other parts.

Finally, all of the calculation zone and domain are meshed by tetrahedral meshes as shown in Figs. 2, 3.

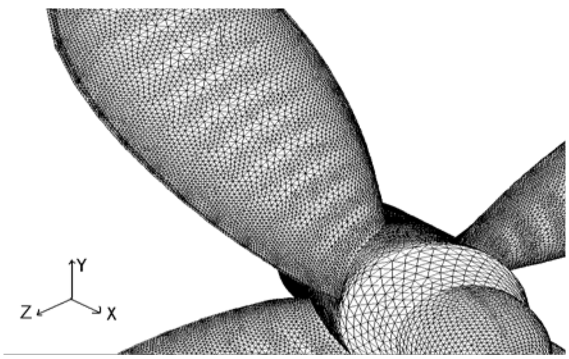


Fig.2: Final mesh for the calculation

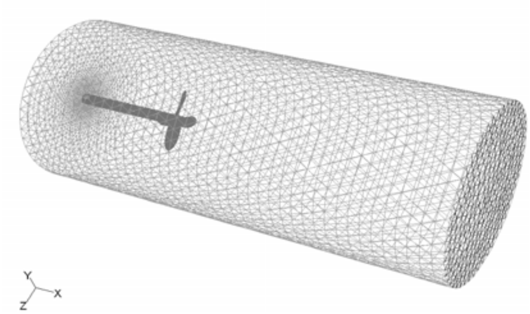


Fig.3: Perspective view of the domain

2.3 LINEARIZATION METHOD:

There are two methods for linearization the equations:
-implicit method
-explicit method
For simulating the equations we used the segregated and implicit method.

2.4 DISCRETIZATION METHOD:

There are several methods that depended on the type and accuracy of problems for discretizing the equations:

1- First order upwind
2- Second order upwind
3- Power law
4- Third order QUICK method

We have used third order QUICK method for discretization of the momentum equation and second order upwind method for discretization of the other equations.

2.5 INTERPOLATION METHOD OF THE PRESSURE:

There are several interpolation methods for pressure in the segregation method as follows:
- Standard method
- Linear method
- Second order method
- Body force
- Presto method

The standard method is popular. So we have used the standard method for interpolation of pressure.

2.6 PRESSURE-VELOCITY COUPLING METHOD:

There are three methods for pressure-velocity coupling in the segregation method:

- SIMPLEC
- SIMPLE
- PISO

The algorithm of PISO is applicable in transient flow but SIMPLEC and SIMPLE algorithms are applicable in steady state flow, therefore we have used the SIMPLE algorithm for pressure-velocity coupling.

2.7 MULTI-GRIDS METHOD:

In FLUENT there are two options for increasing the solution speed:
- Algebraic multi-grids (AMG)
- Full approximation storage (FAS)

We must select the AMG algorithm in FLUENT, because we solved our equations by segregation method.

2.8 INTRODUCTION OF PROPELLER ROTATION:

To simulate the rotational region around the propeller in FLUENT, including the Coriolis acceleration in the RANS equations, we have two options:
- MRF (moving reference frame method)
- SLD (sliding mesh method)

The above methods were investigated with several turbulence models for selecting the best method of propeller simulation [2].

2.9 TURBULENT SIMULATION:

There are several methods for turbulent flow simulation and determination, e.g. the Reynolds stress model. These models are divided to three divisions:

-zero equation model
-One equation model
-Two equation model

The zero equation models are solved by algebraic equations. The one and two equation models are used by one and two extra PDE, respectively. Selection of the turbulence model depend on:

- Physics of flow
- Accuracy range
- Computation facilities (RAM, CPU, etc.)
- Duration time for solution

There are several methods for modeling the flow around the propeller. These are as follows [10]:

- In-viscid Model
- Spalart - Allmaras Model
- Standard K-epsilon Model
- RNG K-epsilon Model
- Realizable K-epsilon Model
- Standard K-omega Model
- SST K-omega Model

2.10 FLOW SIMULATION AROUND THE SOLID WALL:

The solid walls impress on the turbulent flows, severely. This is clear that the solid walls impressed on average flow velocity without SLD mesh condition, thus, simulation the flow near the solid wall has two ways:

- Wall function
- Two equation models with low Reynolds Number

The main problem of wall functions is limitation of using in various flows, but in high Reynolds No. flow fields, we could use the wall functions because the physical variable parameters near the wall, need not to solve with CFD. Wall functions are commonly used in industrial simulation.

Y+ is normal orientation on blade surface for description of the flow condition that it is within the range of 50 to 500 [Fig.4], often the flow has high Reynolds No. and the viscid region of tip blades is very narrow, therefore we simulated the flow behavior near the tip blade by wall function.

2.11 CONVERGENCE TEST:

There is no law for general convergence test. We can use two criteria convergence tests for accuracy the tests:

-Mathematical criteria test: convergence of residual conservation equations to certain quantity (10^{-4}).

-Physical criteria test: uniformly and fluctuation of momentum and drag force. [9]

Using the both criteria together is suitable for convergence test in CFD results.

Histories of convergence results the flow field around the propeller has been shown in Fig 5.

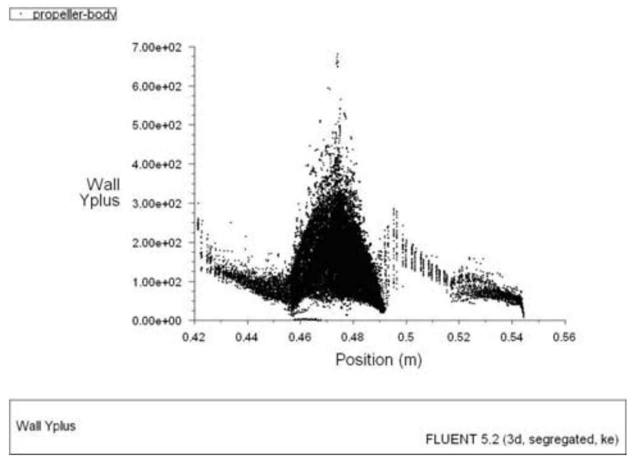
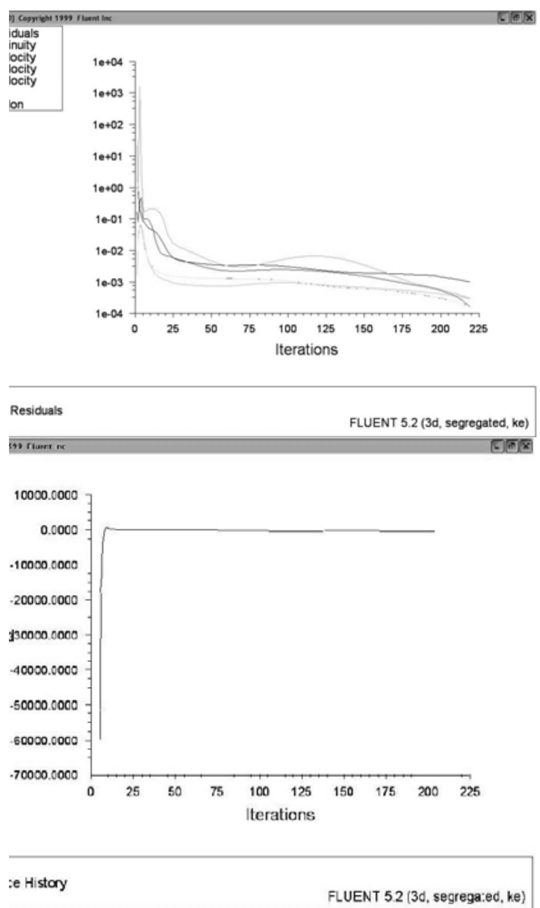


Fig.4: distribution of Y+ on blade surface



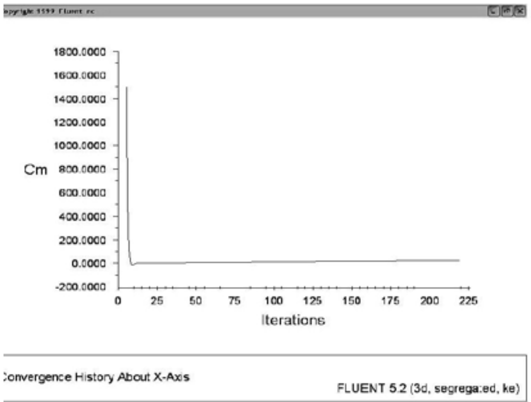


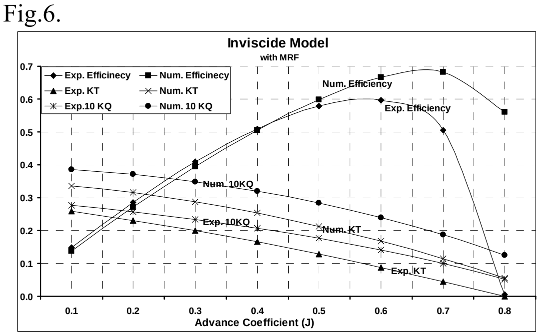
Fig. 5: Convergence history for V_a=1.650m/s, n=900rpm with standard-epsilon turbulence model

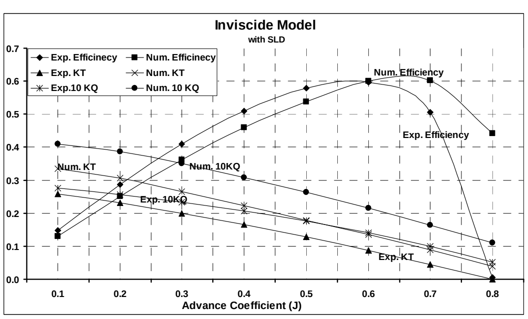
3 PROPELLER PERFORMANCE GRAPHS AND EXPERIMENTAL RESULTS:

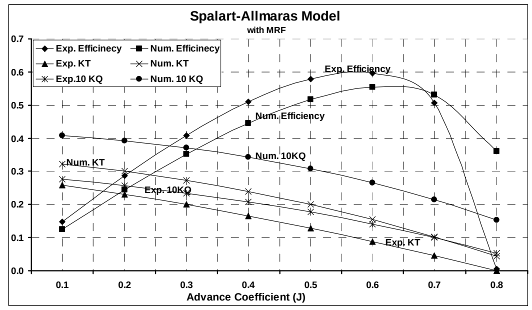
The aim of this paper to model the propeller performance curves by CFD and compare them with experimental results.

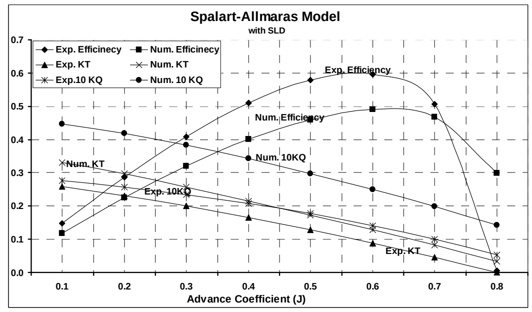
The rotational speed of propeller is constant (n=900 rpm) and we change the inlet velocity of flow between 0.4125 m/s and 3.3 m/s in order to reach the advance coefficient between J=0.1 and J=0.8 .

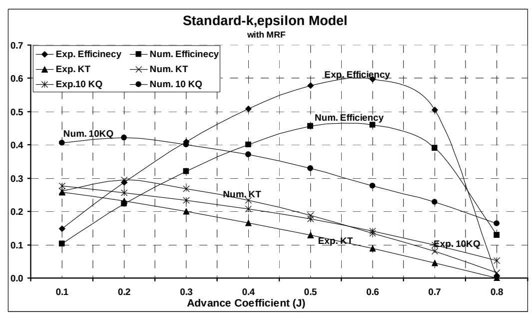
The Computed values of thrust and torque reached the measured values for each velocity. The rotational flow fields around the propeller have been simulated by MRF and SLD methods as shows in Fig.6.

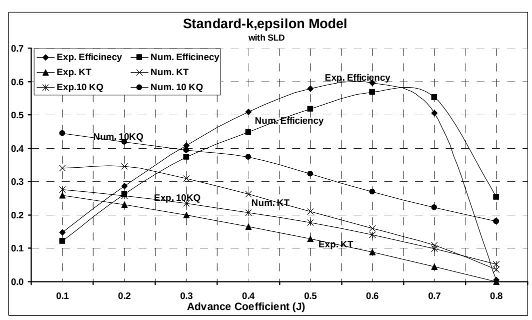


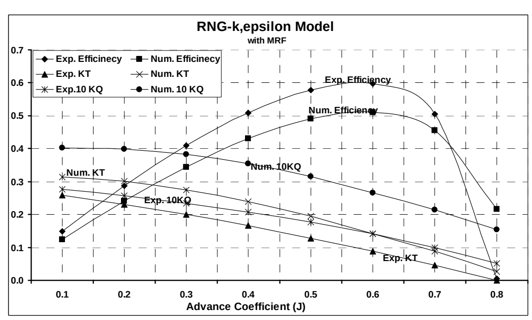


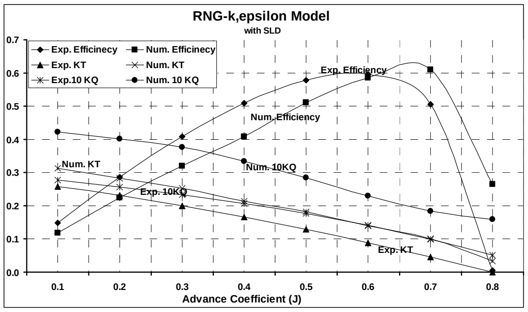


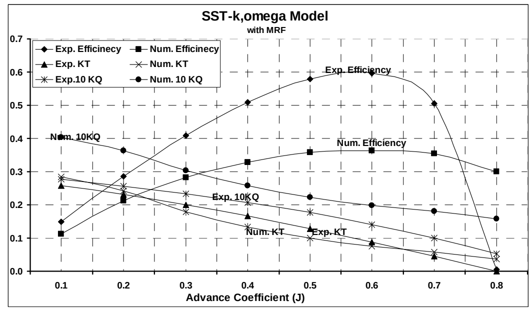


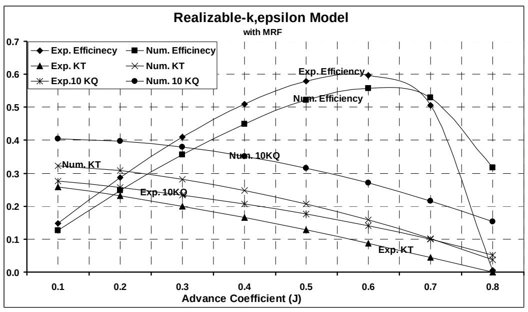


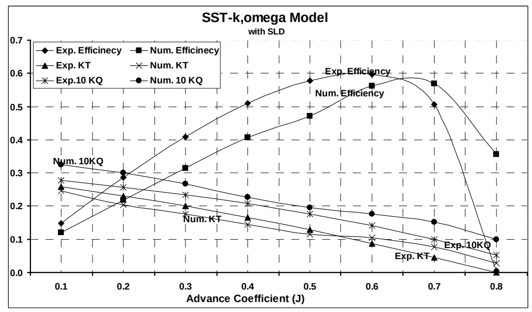


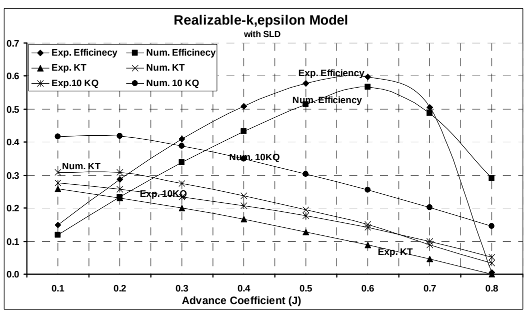


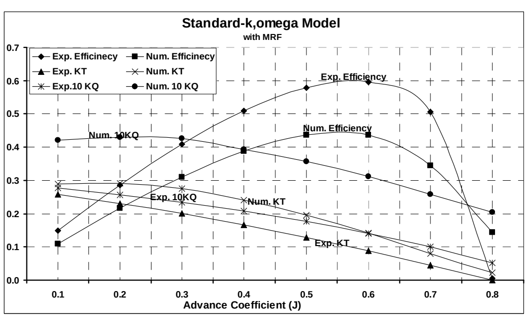












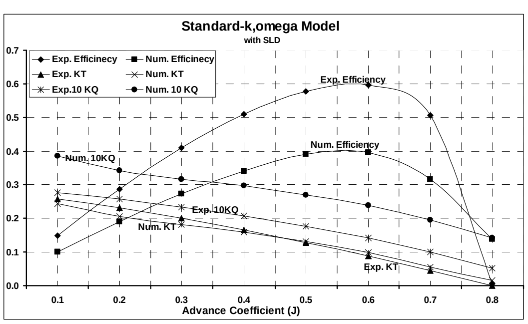


Fig.6: simulation with various turbulent flow models by MRF and SLD methods

The turbulent flow model of SST-k,omega with SLD technique appear to yield the best numerical solution of flow around the propeller.[Fig.7]

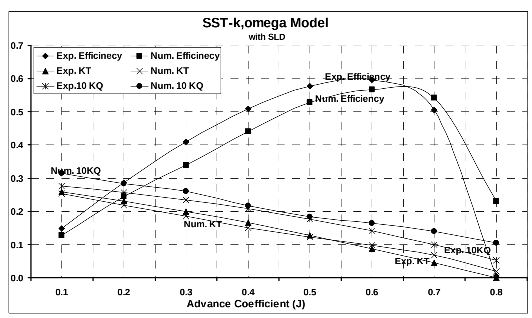


Fig.7: Simulation of turbulent flow model by SLD technique

3.1 INVESTIGATION OF VELOCITY AND PRESSURE FIELD AROUND THE PROPELLER:

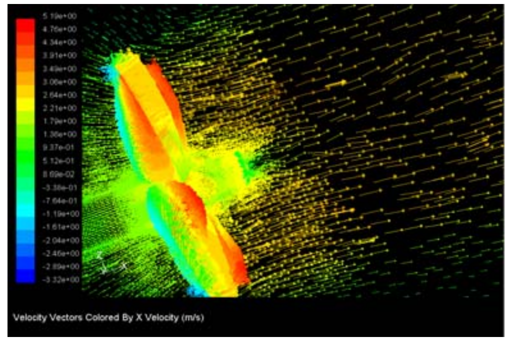
The theoretical flow behavior around any propeller blade is often calculated and explained using the momentum theory [4]. According to this theory, the pressure in the fluid will suffer a continuous decrease as it approaches the propeller, due to the suction generated. In parallel, with the pressure decrease, the speed of the fluid will increase as it comes closer to the propeller. This pressure decrease is kept up to the propeller itself, where an abrupt pressure increase happens due to the tangential shear produced by the blades.

Thus, the axial velocity component has relatively small values in the vicinity of the blade tip and propeller hub. Velocity deficit dominates the region around the blade tip [11].

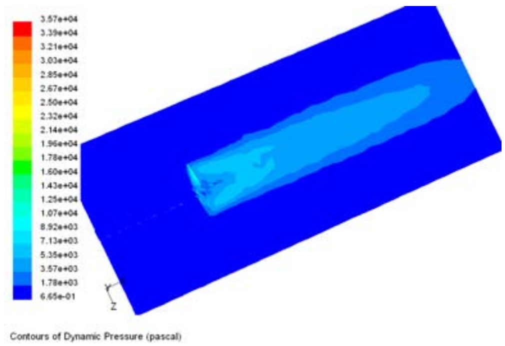
For verification of the numerical simulation the pressure and velocity fields around the propeller have been obtained. The pressure and velocity contours around the propeller in J=0.1 and J=0.8, are shown in Fig. 8

The flow simulation results are verified by considering the contour results.

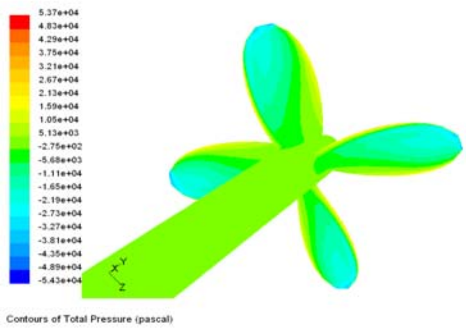
The propeller performances influence the flow field around it decreases with increase in the advance coefficient (J), also the propeller does not influence the pressure and velocity profiles, behind and front of propeller.



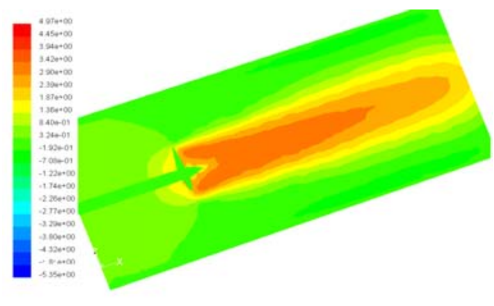
Contours of x velocity in J = 0.1



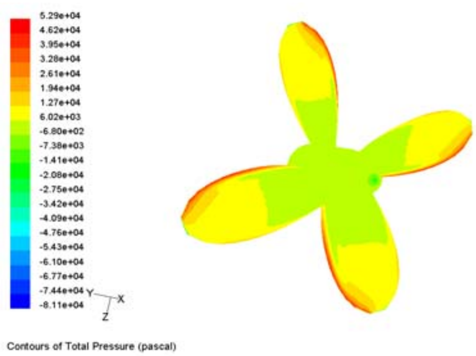
Contours of dynamic pressure in J = 0.1



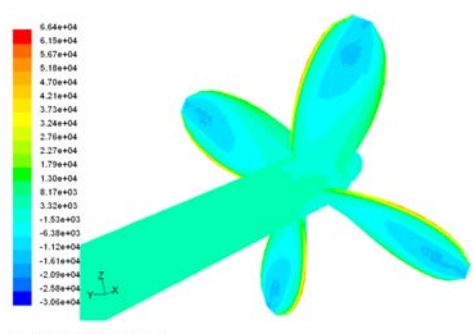
Contours of total pressure on suction surface in J = 0.1



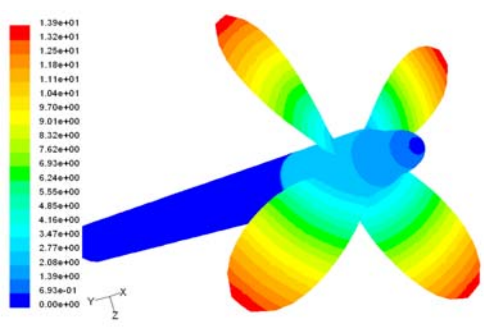
Contours of x velocity in J = 0.1



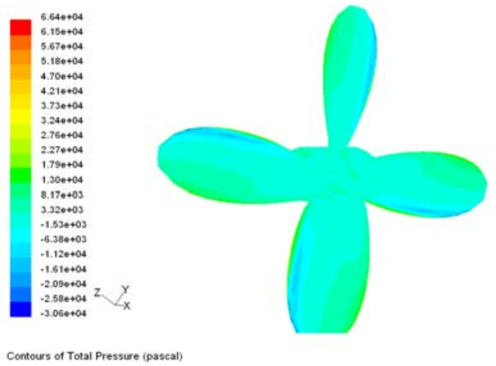
Contours of total pressure on pressure surface in J = 0.1



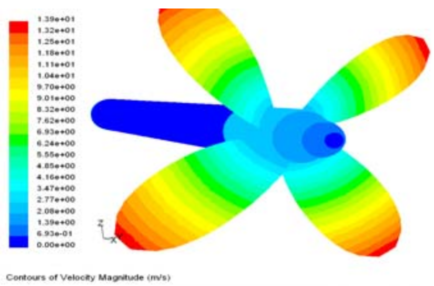
Contours of total pressure on suction surface in J = 0.1



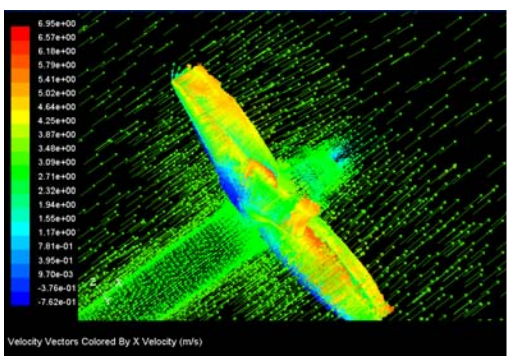
Contours of velocity magnitude on surface in J = 0.1



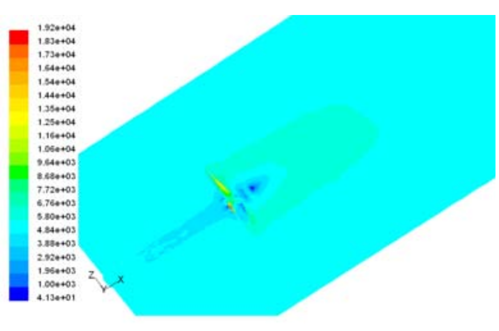
Contours of total pressure on pressure surface in J = 0.1



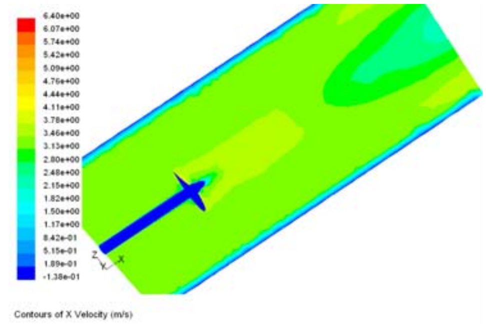
Contours of velocity magnitude on surface in J = 0.1



Velocity vectors by x velocity in J = 0.8



Contours of dynamic pressure in J = 0.8



Contours of x velocity in J = 0.8

Fig.8: contours of pressure and velocity around the propeller

3.2 COMPARISON BETWEEN EXPERIMENTAL AND THEORETICAL RESULTS:

For assessment of marine propeller hydrodynamic performance, we have used the SST-k, omega turbulence model with SLD technique so that its numerical results are near the experimental results.

In error bar of numerical results, for thrust coefficient force (KT) and torque coefficient (KQ), average of values are around 8 and 13 percent, respectively. Prediction results of error in KQ is greater than that. [Figs.9, 10]

In Fig .11 we can see the prediction results of error in propeller efficiency (η), around 11 percent, and the errors are decreased with increasing the advance coefficient (J).

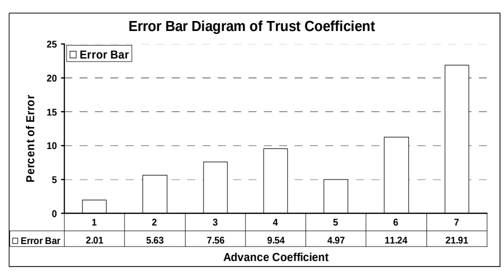


Fig. 9: Error bar for prediction of K_T

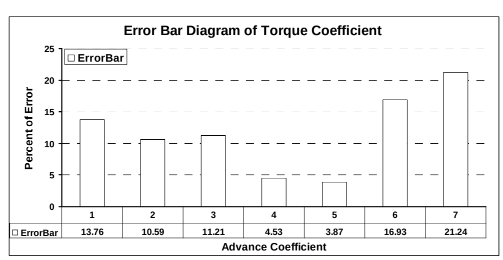


Fig. 10: Error bar for prediction of (K_Q)

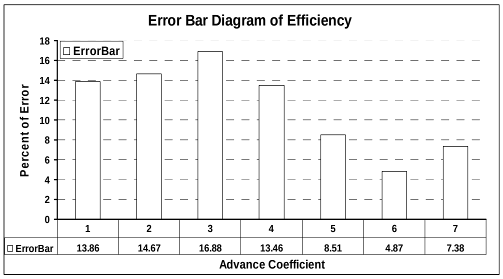


Fig. 11: Error bar for prediction of η

3.3 THE WAYS OF APPROACHING TO ACTUAL CONVERGENCE ANSWERS ACCORDING TO PHYSICS OF FLOW:

We can approach to the suitable convergence answers by considering:

- Boundary condition
- Mesh density and spatial distribution
- Solution procedure

3.4 INSTABILITY SOURCES OF SOLUTION:

Instability show itself, by nonphysical quantities and negative k, ε data. Some instabilities show themselves as not actual data of length scale and time scale in turbulent flow.

Causes of these instabilities are in k, ε equations; therefore it must be accurate selection of turbulent parameters.

3.5 STABILIZING TECHNIQUES:

Two techniques can be used for stabilization and decreasing the instability:

- Up winding
- Clipping

The aim of upwind technique is stabilization of advection terms in high advection zones that using by:

streamline method

first order method

hybrid method

Clipping is the process that it prevents of instabilities in the first and third groups of above mentioned source errors. In this method, it is to determine the positive minimum quantity for k, ε. We can prevent decreasing of the minimum determined quantity.

4 CONCLUSIONS:

The numerical method based on RANS equations is useful and suitable method for analysis of the flow behavior around the propeller .

These results are compatible to the experimental results. The benefits of this method are:

- Decreases time and cost
- Explains the flow around the propeller with pressure and velocity contours,
- No limitation in velocity flow and model size.
- Simulation with simple assumption

ACKNOWLEDGMENTS

The authors gratefully acknowledge the Professor Migeotte and Professor Rad Sharif University of Technology, Islamic Azad University, Bushehr branch for their technical suggestion.

REFERENCES

[1] Gunzburger, M.D., Nicolaides, R.A., “Incompressible computational fluid dynamics”. Trends and advances, Cambridge University Press, 1993

[2] Soltani, M. and Rahimi, R. “computational fluid dynamic by FLUENT software”, Tehran, 2003

[3] Uto, S. and Kodama, Y., “Application of CFD to the Flow Computation around a Marine Propeller –Grid Generation and Inviscid Flow Computation using Euler Equations- ”, J. Kansai Society of Naval Archtects, No. 218, pp. 171-180, 1992

[4] Funeno, I. and et al., “Analysis of Steady Viscous Flow around a Highly Skewed Propeller (in Japanese)”, J. Kansai Society of Naval Architects, No. 231, pp. 1-6, 1999

[5] Funeno, I., “On Viscous Flow around Marine Propellers –HubVortex and Scale Effect-”, Proceedings of New S-Tech 2002 (Third Conference for New Ship and Marine Technology), pp. 17-26, 2002

[6] Martínez-Calle, Julián, “AN Open Water Numerical Model for a Marine Propeller: a Coparsion with Exprimental Data”, Proceedings of the ASME FEDSM’02 2002 Joint US-European Fluids Engineering Summer Conference July 14-18, 2002. Montreal, Canada

[7] Yoshihisa TAKEKOSHI, “Simulation of Steady and Unsteady Cavitation on a Marine Propeller Using a RANS CFD Code”, Fifth International Symposium on Cavitation (CAV2003) Osaka, Japan, November 1-4, 2003

[8] Shin Hyung Rhee *, EvangelosKoutsavdis, “Two-dimensional simulation of unsteady marine propulsor blade flow using dynamic meshing techniques”, Fluent Inc., 10 Cavendish Ct., Lebanon, NH 03766, USA, 2004

[9] Freitas, C,J., “Journal of fluids engineering editorial policy statement on the control of numerical accuracy,” ASME J. Fluids Eng., 115, pp. 339-340. 1993

[10] Shojaeefard, M.H., Noor poor hashtroodi, A.R, “Introduction of computational fluid dynamic”, Tehran, 2000

[11] Paik, B.G. , Lee, C.M. , Lee, S.J. ,”PIV analysis of flow around a container ship model with a rotating propeller”, Experimental in fluids 36 , p833-846 , 2004.