Modeling and Simulation of the Bollard Pull Test on twin propeller tugboats using CFD

David Lopez
Polytechnic University of Catalonia, Spain

Jouse Hernandez
Technological University of Panama, Panama

Carlos Plazaola
Technological University of Panama, Panama

Ilka Banfield
Technological University of Panama, Panama

Adan Vega
Technological University of Panama, Panama

ABSTRACT

A finite volume model had been developed to simulate the bollard pull test of twin propeller tugboats. The Navier-Stokes equations are solved by using ANSYS FLUENT along with the Reynolds Averaged Navier-Stokes and the SST k-w turbulence model. A study is carried out for the standard test conditions using steady and transient methods. In addition, a study is presented for situations where the test environment cannot comply with the ideal conditions of the test. Thus, the effect of dimensions of the test channel was consider. A validation of the results by comparing with experimental and theoretical data was made with very good agreement.

KEY WORDS: Bollard pull; tugboats; twin propeller; propeller thrust; modeling and simulation; CFD;

INTRODUCTION

During the design of a tugboat, one of the bottleneck to be overtaken is the fact that the vessel must produce the required thrust. However, thrust of this type of ship is hard to predict by means of theoretical relationships. To undertake this, since long time ago, ship designers have carried out experiments using scale models in water tank. However, the behavior of bollard pull cannot be reproduced accurately in water tank due to the influence that the water flow has in scale propellers. In order to solve this situation, the bollard pull test was stablished (Verhagen (1970), Japan Workvessel Association (1967, 1972 and 1979) and Kansai Society of Naval Architects (1960)). However, this test requires field instrumentation and the ship presence, thus, if the requirements are not satisfied it will be necessary to take the project to the drawing board with all its consequences (Choi, Min, Kim, Lee and Seo (2010), Mertes and Heinke (2008), Zhang, Hong, Kasilnikov andTang (2008), Carlton (2007), Lee and Sadakane (2007), Lee, Sakai, and Sadakane (2004)).
Computational Fluid Dynamics (CFD) can be used with full advantage for propeller design and related problems in naval engineering such as the bollard pull test (Galeano et al. (2012), Maksoud y Heinke (2002), Tadashi Taketani, Koyu Kimura, Norio Ishii, Masao Matsuura, Yuichi Tamura (2009)). Martínez de la Calle et all (2002) developed two methods: a numerical and other experimental, for the study of flow around a marine propeller. They conducted a flow analysis and presented the characteristic curves dimensionless curves of a marine propeller; obtained both experimentally and by simulation. They used a scale propeller model and after developing methodologies relating to both, numerical and experimental, they were able to achieve very similar results. This indicated the validity of the numerical simulation as a tool for the design and analysis of the flow in a marine propeller. Isao Funeno (2009) also used computational fluid dynamics for the hydrodynamic analysis of azimuthal propellers, considering incompressible viscous fluid. He used turbulent k-ω SST model with wall function. The effect on the flow field was considered introducing propeller centrifugal forces and coriolis forces in a coordinate system relative to the body forces using the average Reynolds formulation of Navier-Stokes equations.
Lam et all (2012) conducted a study employed the average Reynolds of Navier-Stokes (RANS) using computational fluid dynamics to predict the flow of water through a propeller and made a comparison of turbulence models with experimental results obtained. The turbulence models they used were those of the Boussinesq family. Performed the experiment with the model k-ε standard, RNG k-ε, realizable k-ε, k-ω standard, SST k-ω and Spalart-Allmaras. After that they were able to present advantages and disadvantages of one model over the other.
In this paper the authors presents a methodology to model and simulate the bollard pull test of twin propeller tugboats, during the design stages. The aim is to predict the thrust with a low margin of error compared with the experimental results, for different configuration of hull, propeller and nozzle. First, ANSYS FLUENT software, based in finite volume method, is used to study the bollard pull test. The Navier-Stokes equations are solved along with the Reynolds Averaged Navier-Stokes (RANS) and the SST k-w turbulence model are solved using this software. Then, a numerical study is carried out for the standard test conditions using steady and transient methods. After that, a study is presented for situations where the test environment cannot comply with the ideal conditions for the test. Finally, the effect of dimensions of the test channel, depth and marine currents were evaluated and the results compared with experiments.

DEVELOPMENT OF THE FINITE VOLUME MODEL

The finite volume model developed for this study is presented in this section. The mesh model, boundary condition as well the main characteristics of the propeller, the nozzle and the ship hull are presented and explained in details.

Numerical model

A numerical approach to the solution is taken. In this case, it is based on the Reynolds Average Navier-Stokes (RANS), in which the flow variables are divided into two components, a mean component and a fluctuating component. The computational domain has been designed with the objective of describing the ideal environment for the Bollard

Pull test. The domain has a parallelepiped domain with dimensions of 400x200x44 meters. The forward velocity of the vessel is zero for a Bollard Pull Test. Therefore the forward coefficient, given by Equation 1, is equal to zero (J=0).

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

where *Va* is the forward velocity, *n* are revolutions per second and *D* the diameter of the propeller. The thrust coefficient (K_T) and the torque coefficient (K_Q) are given by Equation 2 and 3, respectively:

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

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

where T is the propeller-nozzle thrust force, ρ is the fluid density and Q represents the propeller torque.

Proppeler-hull main characteristics

The propulsion unit is an azimuth propeller with transmission at right angle and nozzle. The propulsive system comprises two units Ulstein Aquamaster (model US-2001/3150). The tug has installed two engines 9L20 WÄRTSILÄ 2200 BHP, which provide a total throw of 52 tons.
The main characteristics of the propeller are given in Table 1.

Table 1. Selected propeller main characteristics

Propeller	Kaplan 4.70
Propeller diameter	2300 mm
Pith ratio	1
Maximum speed	267 RPM
Reduction ratio	3.748:1
Nozzel	19A
Nozzel diameter	2325 mm
Nozzel length	1150 mm

A geometric model for the hull of the vessel was also developed based on the drawing of an existing tugboat. The main characteristics of the ship model are presented in Table 2. Figure 1 shows the ship model including the nozzle propeller arrangement.

Table 2. Selected ship model main characteristics

Draught (m)	2.5
Length (m)	29.84
Beam (m)	11.0

Development of the mesh model

Two domains were created using ANSYS Design Modeler. A stationary domain which includes the fluid, hull, nozzle and the gear box, and a rotational domain that surrounds the propeller. ANSYS Meshing was used to generate the mesh. A non-structured tetrahedral mesh in Patch Independent modality was used, in this modality the mesh is first created on the faces of the domain and on the edges thereafter. In this way skewness asymmetry is reduced and at the same time the mesh quality is improved. For the propeller zone asymmetry of 0.6948 was obtained. Figure 2, the geometry of the stationary domain used in the computation is presented. Figure 3 shows the mesh model of the static domain. It has 8,256,043 elements and 1,492,315 nodes.

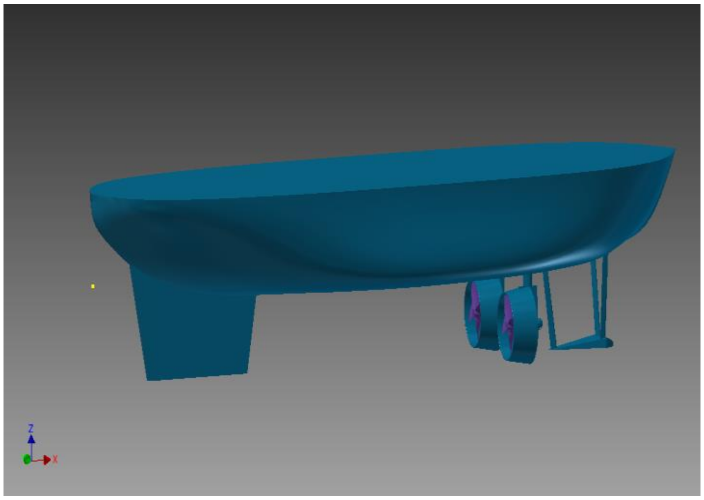


Figure 1. Vessel model used in the simulation

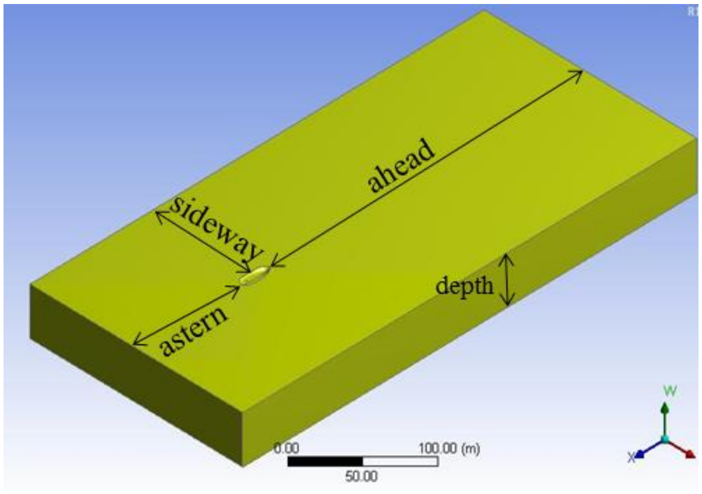


Figure 2. Stationary domain used in the simulation

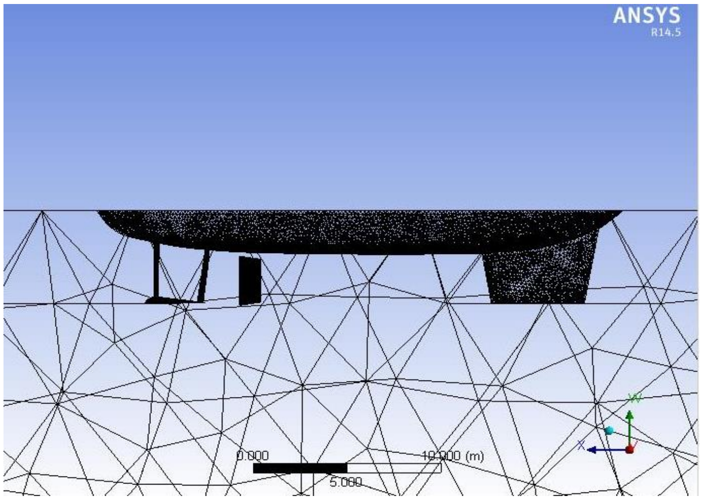


Figure 3. Mesh model of the static domain

Boundary conditions

The boundary conditions using in the simulation of the bollard pull test, under ideal condition, are given in Table 3. As is seen in further sections, different boundary condition were needed in order to perform some specific simulations.

Analysis of the results

Two types of simulation: steady and transient state are performed. In the steady state simulation, The Multiple Reference Frames (MRF) scheme is used. In this scheme, it is possible to define translational and rotational speeds. Thus, a speed has been established for the rotational domain. For a reference frame with constant rotational speed, it is possible to make a transformation of the equations of motion to obtain a solution in a stationary reference frame. At the domains interface, a local reference frames transformation is made so that the flow variables can be used in adjacent zones. The MRF approach does not takes into account the relative motion of one zone with respect to adjacent zones, the mesh remains fixed. Through the calculations, the moving part is frozen in a given position and the propeller flow field is obtained, Zhang et al. (2008). The convergence criteria was established in a residual size of the order of $1x10^{-4}$. A convergence test was also carried out using $1x10^{-5}$, the results were similar to those obtained using $1x10^{-4}$. Examples of the results of the steady state simulation is given in Figures 4, 5 and 6.
For the simulation in transient (unsteady) state, the sliding mesh scheme was used. In this case, the unstable interactions of relative motion are included. In general, the sliding mesh scheme is considered with higher precision but it requires a higher computational demands. First, the steady state solution was used as initial condition. The time step interval was taken as 0.07 seconds and 20 iterations for each interval (time step). A 30 seconds simulation time was taken, since that is the interval used in field tests. The total simulation time was of about 85 hours. The results of transient state are not presented in this paper becouse two main reasons: the results of steady state simulation were good enough, compared with theoritical results, and second, due to the large computational time required for complete a single case when transient state simulation was used.

Table 3. Boundary conditions used in the simulation

Ship element	Type of boundary condition
Hull/Propeller/Nozzle/Gear box	No Slip
Water surface	Symmetry
Bottom	No Slip
Outer boundaries	Hydrostatic pressure

Validation of the finite volume model

As a way of validate the numerical model, simulations were carried out and the results compared with experimental data found in the literature, for the case of the tractor tugboat mentioned above. The results of the comparison are given in Table 4. In this, five cases considering different speeds and channel dimensions are studied. Note that the difference in between the numerical and the experimental results is small, in the order of less than 0.5% except in Case 4, where the computational convergence was not attain, so the thrust could be under predicted (see following section). Base on the comparison, we can say that our numerical model is suitable for studying the bollard pull test. It is important to mention that there is a difference of 3 tons of force between the simulation results and theoretical results. This difference is because the actual performance of two propellers installed on a ship, is not twice the theoretical yield of a propeller studied independently. In the following section, the analysis of the results of these five cases is expanded and discussed in details.

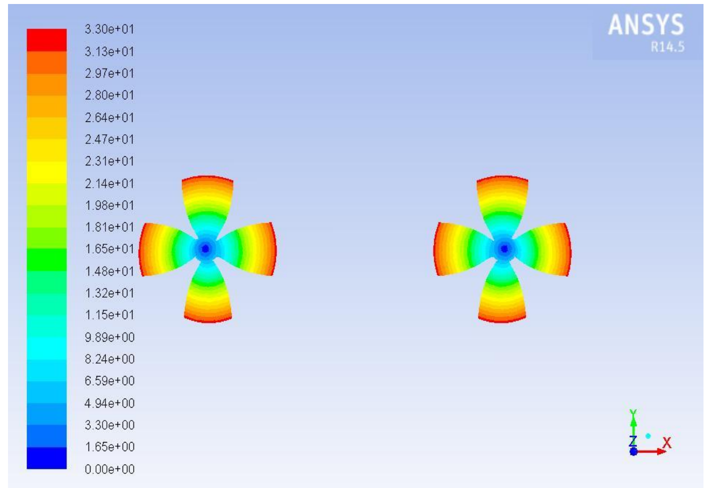


Figure 4. Velocity contours of the propellers

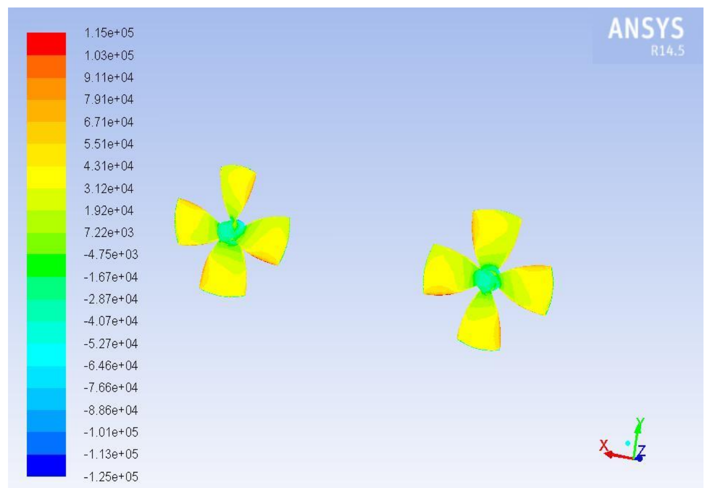


Figure 5. Pressure contour of the suction side

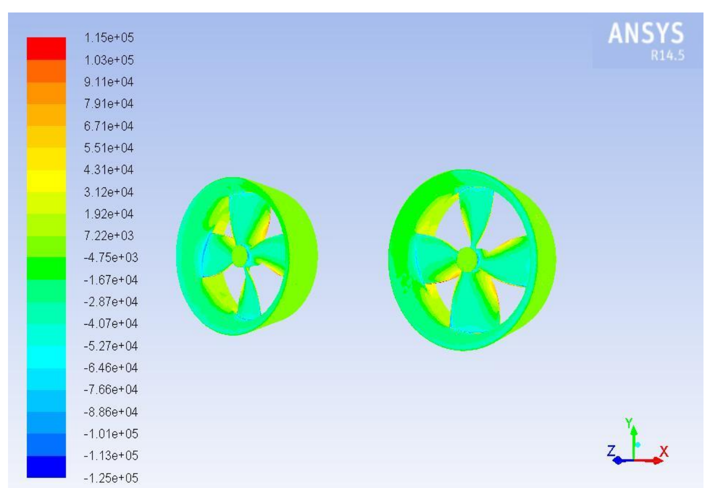


Figure 6. Pressure contour of the suction side (propellers and nozzle)

Table 4. Comparison of numerical and experimental results for a 2 meters diameter propeller

	Case 1	Case 2	Case 3	Case 4	Case 5
Propeller speed (RPM)	267	254	254	254	254
Theoretical thrust (Ton-force)	60.98	55	55	55	55
Experimental thrust (Ton-force)	--	52	52	52	52
Simulated thrust (Ton-force)	58.09	52.39	52.20	48.45	51.76

INFLUENCE OF THE DIMENSION OD THE TEST SITE ON THE BOLLARD PULL TEST

In order to perform the bollard pull test following the rules of classification society, the vessel have to be located in a place that meet the requirements for water depth, width and length of the channel and no less important, the sea currents. Usually it is difficult to meet these requirements, so the test is carried out without following the recommendation. This, of course, implies the possibility of over/underestimate the real capacities of the vessel.

In this section, a study on those influential factors affecting the bollard pull test is presented. The objective of this comparison is to study the influence that the dimension of the channel (See Figure 2) has on the results of the test. The dimension used are based on the recommended rules provided by the classification society. Five cases are studied. The dimensions of the channel are varied in order to reproduce the possible situation, from this, five cases results as follows:

Case 1. Bollard pull test in open water at maximum power
Case 2. Bollard pull test in open water at operation power
Case 3. Bollard pull test under classification society rules
Case 4. Bollard pull test under minimum condition permitted by the classification society and
Case 5. Bollard pull test under the minimum requirement of depth and astern distance permitted by the classification society.
Case 6.

Table 5. Dimension of the channel for each case (See Figure 2)

Position from the reference	Distance in meters				
	Case 1	Case 2	Case 3	Case 4	Case 5
Ahead	127	127	127	127	127
Astern	300	300	300	55.8	55.8
Sideway	100	100	100	100	100
Deepth	40	40	20	13.31	13.31

Table 6. Boundary condition for each case (See Figure 2)

Distance (m)	Boundary Conditions				
	Case 1	Case 2	Case 3	Case 4	Case 5
Forward	Pressure outlet	Pressure outlet	Pressure outlet	Pressure outlet	Pressure outlet
Backward	Pressure outlet	Pressure outlet	No slip wall	No slip Wall	No slip wall
Sideways	Pressure outlet	Pressure outlet	Pressure outlet	No slip wall	Pressure outlet
Deepth	No slip wall	No slip wall	No slip wall	Noslipwall	No slip wall

Table 5 shows the dimensions of the channel and Table 6 the boundary conditions for each case. Table 7 resume the resulting thrust obtained after simulating each of the five cases. For simplicity, the thrust forces are given for each component of the vessel separately.

Bollard pull test in open water at maximum power

In this case, the force exerted by the tug during the bollard pull test in open water at maximum power was studied. Open water means that there is not any element that influence the results. Since the simulation is at full power, each propeller revolutions are set to 267 rpm. The theoretical force for this speed is of 60.98 Newton Force Tons. In this simulation, the results can only be compared to the theoretical results, since there are no test result of the bollard pull in open water for this specific vessel. As it is shown in Table 7, a total thrust of 58.09 T is obtained, which is 4.73% smaller than the expected theoretical value. Figure 7a and b show the contours of turbulent viscosity ratio produced in the surface and the bottom of the channel during the simulation, respectively. As seen, it is an orderly and uniform wake on the water surface, thus, the turbulence is very low. This low turbulence means that not external influence on the resulting thrust exist when the test is performed in open water.

Table 7. Comparison of the resulting thrust forces for each case studied

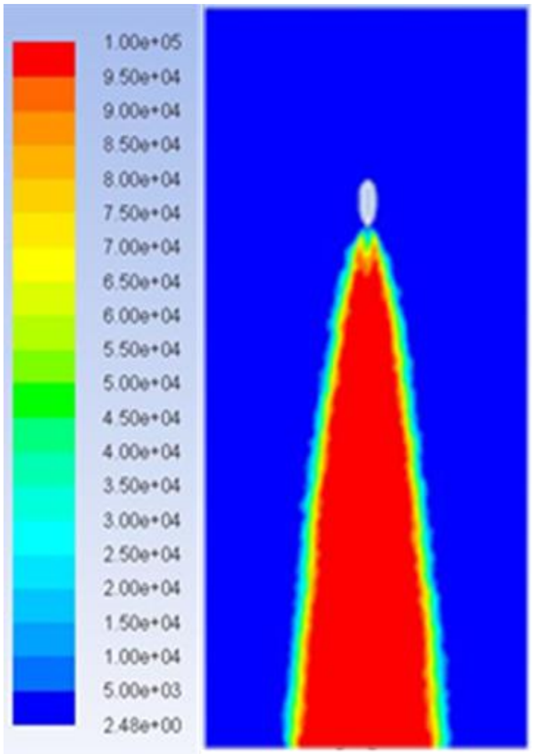
Ship element	Thrust (Ton-force)				
	Case 1	Case 2	Case 3	Case 4	Case 5
Hull	-1.65	-1.49	-1.49	-1.10	-1.37
Nozzle (PRT)	14.23	12.84	12.73	10.96	12.55
Nozzle (STBD)	14.25	12.85	12.79	11.09	12.56
Propeller (PRT)	14.73	13.29	13.27	12.96	13.23
Propeller (STBD)	14.74	13.29	13.29	13.01	13.20
Full vessel	58.09	52.39	52.20	48.45	51.76

Bollard pull test in open water at operation power

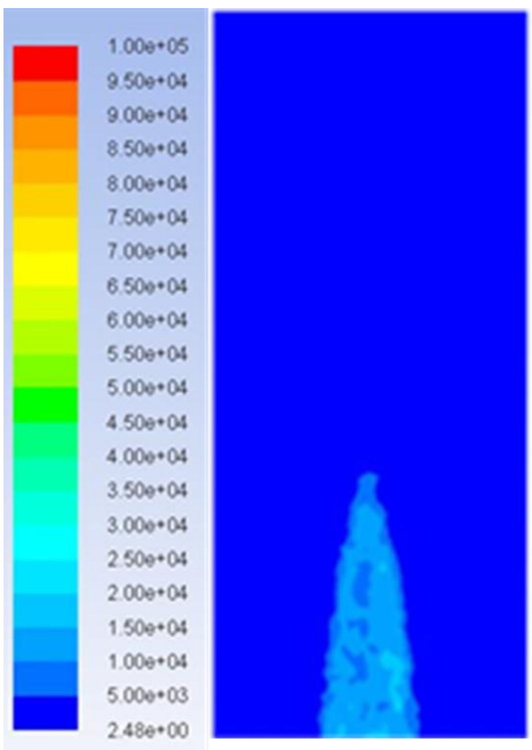
For safety reasons ships does not operate at maximum speed, so the operation power is smaller than the maximum. In order to capture this behavior, simulation of the bollard pull test, at the operation power was also performed. The condition of the test are the same that the previous case as shown in table 5 and 6. The operation speed was assumed to be 5% smaller than the maximum (254 rpm). The theoretical thrust is estimate as 55 tons. The thrust obtained from the simulation (given in Table 7) is of 52.39 tons which is very close to the experimental value. The contours of turbulent viscosity ratio produced in the surface and the bottom of the channel during this simulation is similar to those showed in Figure 7a and b. By comparing the experimental and the simulation results we can conclude that there is not external influence affecting the bollard pull test in this case as it was in previous case.

Bollard pull test under classification society rules

This case corresponds to the equivalent of the actual test that has undergone the tugboat. The speed parameters remain the same as previous case. In order to perform this simulation, the boundary conditions and dimensions of the stationary domain were varied (Table 5 and 6). The difference between the total thrust obtained in the simulation and that obtained during experiments is of 0.20 Tons of Force. This value is 0.38% larger than the experiment. Figure 8a and b shows the contours of turbulent viscosity ratio produced in the surface and the bottom of the channel during the simulation. As seen, even when large turbulence exist in the aft part of the channel and in the bottom, this is far from the vessel so it does not affect the results of bollard pull test.

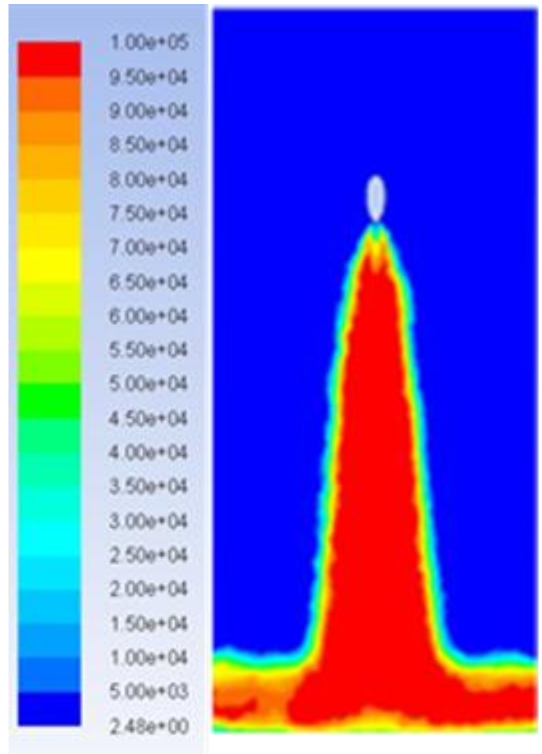


(a)

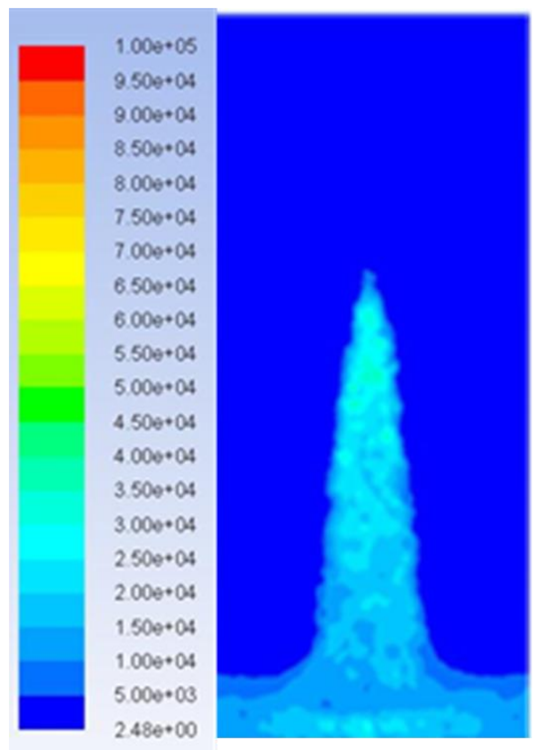


(b)

Figure 7. Contour of turbulent viscosity ratio produced during the simulation of cases 1 and 2: (a) top surface and (b) Bottom surface



(a)

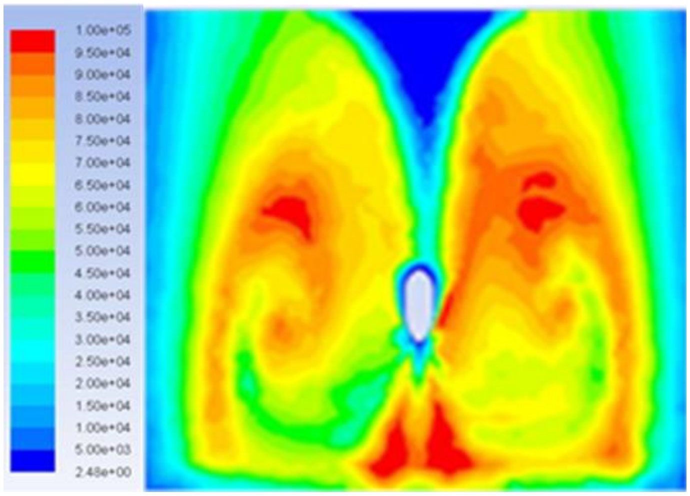


(b)

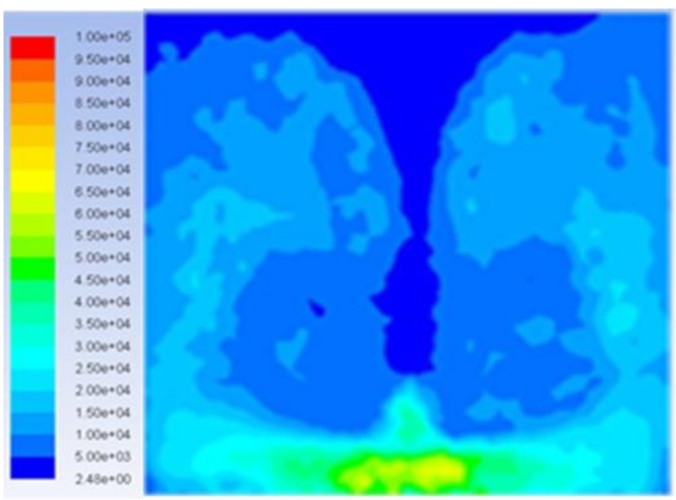
Figure 8. Contour of turbulent viscosity ratio produced during the simulation of case 3: (a) top surface and (b) Bottom surface

Bollard pull test under minimum requirement of depth, astern and sideway distance permitted by the classification society

In this simulation, the bollard pull test is simulated using the minimum values of the draft, the distance astern of the ship and the minimum distance from the ship side to the border (see table 5). Those are the minimum values accepted by the classification society. In practice, a bollard pull test on a site that meets these minimum three conditions simultaneously cannot be accepted, since the results would be much lower than the real pull capacity of the ship. However, our main goal is to be able to accurately simulate the bollard pull thus all the possible condition are considered.



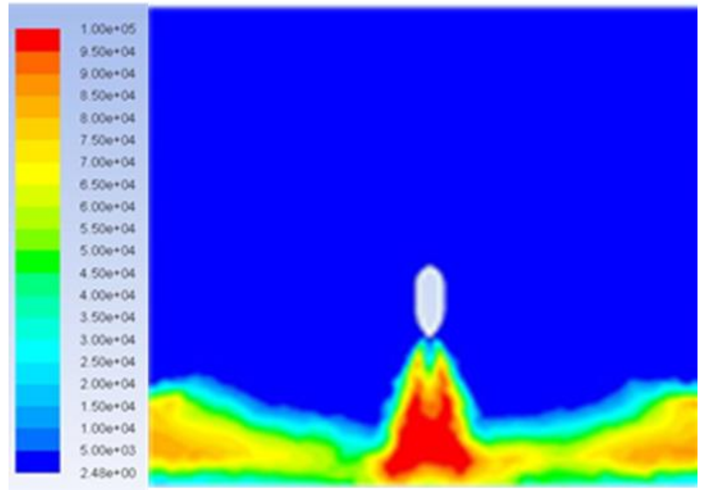
(a)



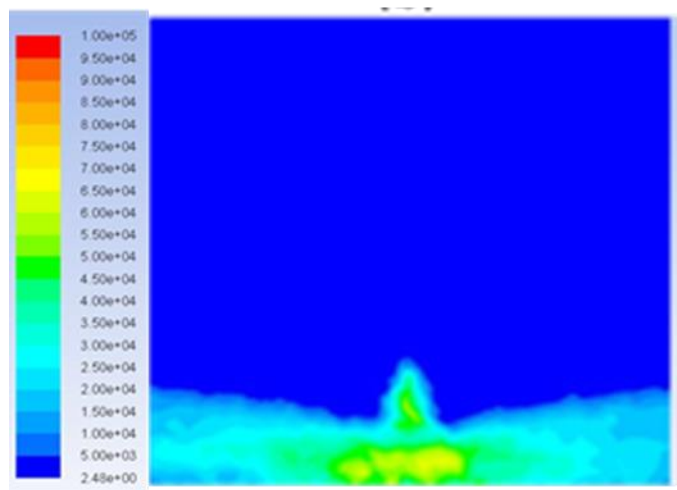
(b)

Figure 9. Contour of turbulent viscosity ratio produced during the simulation for case 4: (a) top surface (b) Bottom surface

The speed and power condition are kept the same as previous case. The dimension of the site and boundary condition are given in table 5 and 6. Figures 9a and b show the contour of turbulent viscosity ratio produced in the surface and the bottom of the channel during this simulation. Noted that the turbulence around the ship is very large. In CFD, when turbulences like the one shown in the figures occur, the convergence of the program become a bottleneck. In our case we could not achieve the final step of the simulation, thus the final thrust was under estimated as is shown in Table 4 and 7. Comparing with experiments, almost 4 tons of difference exists. Even when the result of this simulation is not correct, we consider this case important since the use of CFD is become most popular every day, additional work need to be done in order to undertake this kind of problem.



(a)



(b)

Figure 10. Contour of turbulent viscosity ratio produced during the simulation of case 5: (a) top surface, (b) Bottom surface

Bollard pull test under the minimum requirement of depth and astern distance permitted by the classification society

Due to the non-convergence of the case 4, we repeated the same configuration but varying the boundary condition of the lateral boundaries, as shown in Table 6. Thus, the test in a zone with the minimum draft and free space astern given by the classification societies is simulated. As seen in Table 4 and 7, the thrust obtained is smaller than that of case 3. This shows that when the minimum condition allowed by the classification society are used, the thrust of the tugboats is under estimated. Figure 10a and b show the contour of turbulent viscosity ratio produced in the surface and the bottom of the channel during the simulation. As seen in the figures, the wake is dissipate by the side of the domain after impinging on the quay wall. In addition, the turbulence at the bottom follows the pattern of turbulence of the surface and it is considerably higher than in previous cases, where the depth is greater. Under this conditions, not computational trouble occurred so the results are correct.

CONCLUSIONS

A methodology for analysis and simulation of Bollard Pull Test of twin propeller tugboats, taking into account the interaction propeller-nozzle-hull, has been successfully established. The methodology was validated comparing the results with experimental data reported in the literature. The main conclusion drawn from this study are as follows:

1. The thrust force was determined for simulated ideal test conditions. Unsteady and steady state simulations were carried out and a comparison of computational time and precision of the results leads to conclude that the steady state solution shows good precision and shorter computation time.
2. The thrust force is affected when the test is performed in a place smaller than the recommended by the classification societies, been the free distance from the side of the ship, that from the aft of the ship and the depth of the channel, values that need to be followed, according the regulation, when performed the test.
3. The theoretical thrust in twin propeller tugboats is larger than the one obtained by experiments or simulation. This is because the theoretical thrust is calculated individually for each propeller and then multiplied by the number of thrusters. In practice, the operation of each propeller negatively influences the performance of the other. Therefore, in simulation of the bollard pull test it is not recommended to base the comparison on the theoretical thrust.

ACKNOWLEDGEMENTS

The author wouldlike to thanks to SENACYT Panamá and CLASS IBS for all their support during the development of this research project.

REFERENCES

Abdel-Maksoud, M., Heinke, H.-J.. Scale Effects on Ducted Propellers. 24th Symposium on Naval Hydrodynamics. Fukuoka, Japan. 2002

Galeano Vasconcelos, Leonel; Prieto Fernández, Alejandro P.; Recarey Morfa, Carlos Alexander. Experimentos Numéricos CFD en Propulsión Naval. VII Symmtechnaval 2012.

Isao Funeno, Hydrodynamic Optimal Design of Ducted Azimuth Thrusters. First International Symposium on Marine Propulsors smp'09, Trondheim, Norway, June 2009.J.E. Choi, K. S. Min, J. H. Kim, S. B. J. Japan Workvessel Association (1967). "An Investigation on Harbour Tugboat Assisting for a Large Vessel", The Japan Workvessel Association Report, Japan.

Japan Workvessel Association (1972): "An Investigation on Harbour Tugboat Assisting to Brake a Large Vessel", The Japan Workvessel Association Report , Japan.

Japan Workvessel Association (1979): "The Research on Instruction of Design for Towing Force of Tugboat", The Japan Workvessel Association Report, Japan.

J. Martínez de la Calle, J. González Pérez, L. Balbona Calvo, E. Blanco Marigorta, Análisis del flujo en una hélice marina. Comparación de los resultados numéricos con las medidas experimentales. 2002.

Kansai Society of Naval Architects (1960): "Manual of ship design on Instruction of Design for Towing Force of Tugboat", Published by KAIBU-DO Company, Japan. M. Verhagen, Hoogleraar in De Afdeling der Technische Natuurkunde Voor Een Commissie Uit de Senaat te Verdedigen Op Donderdag 25 June 1970 te 14.00 Uur.

Lee, H. W. Seo. Resistance and propulsion characteristics of various commercial ships based on CFD results. Hyundai Heavy Industries, Lee, S.S. and SADAKANE, H. (2007): "Experimental study on the influence of the propeller immersion depth on towing force of tugboats in regular waves", the Journal of Japan institute of navigation, Japan, No.116, pp.129-136.

Lee, S.S., Sakai, Y. and Sadakane, H. (2004): "Model Experiments on the Tow Force of a Single Propeller Tugboat in Regular Head Waves", the Journal of Japan institute of navigation, Japan, No.111, pp.33-40.

Jeonha-dong, Dong-gu, Ulsan 682-792, Republic of Korea. 2010.

Paul Mertes, Hans-Jürgen Heinke, Aspects of the Design Procedure for Propellers Providing Maximum Bollard Pull. Suntec Convention Centre, Singapore Organised by the ABR Company Ltd. 2008.

S. Carlton. Marine Propellers and Propulsion, Second Edition. Global Head of Marine Technology and Investigation, Lloyd's Register. ButterworthHeinemann. USA 2007.

Tadashi Taketani, Koyu Kimura, Norio Ishii, Masao Matsuura, Yuichi Tamura. Advanced Design of a Ducted Propeller with High Bollard Pull Performance. First International Symposium on Marine Propulsors. Smp'09, Trondheim, Norway, June 2009.

W.H. Lam, D.J.Robinson, G.A.Hamill, y H.T.Johnston, An effective method for comparing the turbulence intensity from LDA measurements and CFD predictions within a ship propeller jet. 2012.

Zhang Zhi-Rong, Hong Fang-Wen, Vladimir Kasilnikov, Tang Deng-Hai, Progress in Analysis of Viscous Flow around Podded Propulsor. China Ship Scientific Research Center, Wuxi, Jiangsu, China, and MARINTEK, Trondheim, Norway. 2008.