Second International Symposium on Marine Propulsors
smp'11, Hamburg, Germany, June 2011
Workshop: Propeller performance

Open Water Test for Testing Methodology

Alejandro Caldas[1], Marcos Meis[1] , Adrián Sarasquete[1]

[1]Vicus Desarrollos Tecnológicos (VICUSdt), Vigo, Spain

1 INTRODUCTION

The main goal of the test 2 of workshop is to determine whether our industrial field methodology is able to reproduce the results from experimental test. This methodology obeys the same one described in "Caldas et al (2010)" but in this time, the values of *y+* are close to 1. In this paper, an Open Water tests for the propeller have been carried out at different advance ratios and a mesh convergence study.

2 GENERAL DESCRIPTION

In the following lines, the descriptions of the numerical model and calculation procedure are shown.

2.1 Numerical method

The mathematical model used for the calculation of the numerical simulations is described by Reynolds Averaged Navier Stokes Equations (RANSE). We get the couple of the Reynolds shear stress tensor by means of a two-equation model named Two Layer Realizable K-Epsilon using an All y+ Law for the wall modelling.

Commercial code StarCCM+ has been used for the numerical solutions of the equations. StarCCM+ solves RANSE equations in their integral form, by means of Finite Volumes methods. The numerical method uses second-order schemes both for viscous and convective terms. Velocities and pressures are solved in a segregated manner, and then coupled by means of the SIMPLE algorithm. The discrete equations are solver using pointwise Gauss-Seidal iterations, and algebraic multi-grid methods accelerates the solution convergence ("STARCCM+ (2009)").

The rotation of the propeller as modelled used a moving reference frame system.

Finally, just said that the free surface has not been modelled making both modelling and resolution easier.

2.2 Computational domain and grid details

For the simulation, the computational domain was created as one passage surrounding a blade. The domain presents a the following dimensions: inlet a 4D upstream; exit at 10 D downstream; outer boundary at 4.5D form the hub axis.

All the surfaces were meshed with triangles. The inner region was made on a non-structured mesh of polygons. The layer thickness was meshed with prism and its size was sufficient refined to ensure y+ values close to 1 toward foil surface. In the regions which present a small gap, the prism layer thickness was disable.

Three different volume shapes were created close to the blade to define regions with a continuous increment on the size of mesh, giving rise up to smooth transition on the grow of mesh size from blade.

The number of polygons was about 560,000 for the parametric study. In the case of mesh convergence study, three different meshes were employed with a constant growth (with a value of sqrt(2)) from the initial mesh. The number of polygons were about: 180,000, 560,000 and 1,200,000

2.3 Boundary conditions

The problem is closed establishing the initial and boundary conditions on the physical and computational boundaries. A no-slip condition is imposed on the solid walls (velocity = 0; normal pressure gradients = 0). A condition of velocity is imposed upstream, whereas downstream, there is a pressure condition. Regarding the initial conditions two types were established: for the convergence study, previous solutions were used as initial conditions so as to accelerate convergence; for the parametric study a condition of constant velocity over all the domain, equal to the inlet velocity.

Two rotationally periodic boundaries (interfaces) with 72º angle in between were used to model the rotational symmetry present in the geometry.

2.4 Convergence study

As unstructured grids are employed, the convergence study was carried out according to integral values of thrust and torque by an error. This error computed as the difference between integral values for a mesh respect to the finest one.

The figures of merit used to establish the comparison among the studies correspond to the following adimensional standard coefficients:

$$K_T = \frac{T}{\rho \cdot n^2 \cdot D^4} \quad K_Q = \frac{Q}{\rho \cdot n^2 \cdot D^5} \quad \eta = \frac{K_T \cdot J}{K_Q \cdot 2\pi}$$

where Kt= the thrust coefficient, Kq= the torque coefficient, and η = the efficiency.

The convergence criteria in the present study were at least a four orders of magnitude drop in the mass conservations imbalance and momentum equations residuals, which are deemed sufficient for most steady flow solutions.

3 MESH CONVERGENCE STUDY

As it was mentioned before, the convergence study was performed by errors of thrust and torque coefficients on the different meshes. This mesh convergence study was carried out for a advanced ratio of 1,4. The values of coefficients and the values of these errors are shown in the table 1.

Table 1. Adimensional coefficients and errors for J=1,4

Mesh	Computed		Error (%)	
	KT	KQ	KT	KQ
Mesh 1	0,0326	0,0112	2,20%	1,57%
Mesh 2	0,0323	0,0112	1,33%	1,17%
Mesh 3	0,0319	0,0110	0,00%	0,00%

As it can be seen at table 1, the error of mesh 2 is close to 1%, and therefore, this mesh is used as mesh for the parametric study.

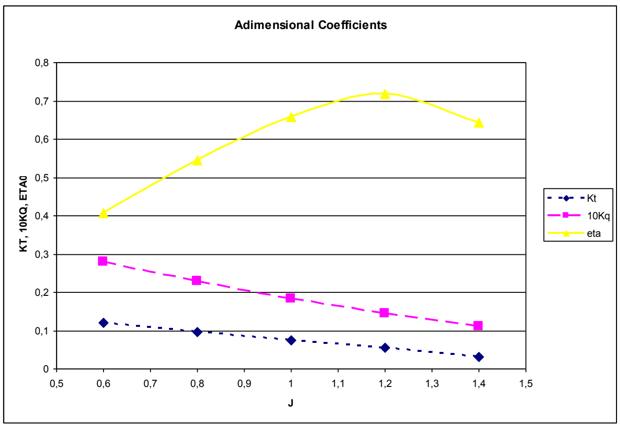
4 PARAMETRIC STUDY

The values of parametric study are shown in table 2 and plotted on figure 1.

Table 2. Adimensional coefficients of parametric study

Computed values			
J	KT	KQ	Eta
0,6	0,119	0,028	0,407
0,8	0,097	0,023	0,545
1,0	0,076	0,018	0,659
1,2	0,055	0,015	0,717
1,4	0,032	0,011	0,644

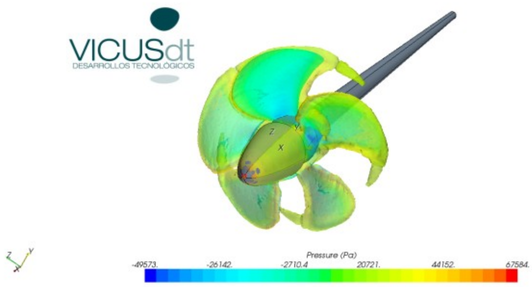
Figure 1. Adimensional coefficients of parametric study



The maximum efficiency is obtained at values of advanced ratio close to 1,2.

In figure 2, an image of vorticity near the propeller is shown. The vortex shedding behind the blade can be observed perfectly.

Figure 2. Vortex shedding from propeller



5. CONCLUSION

An Open Water tests for the propeller have been carried out at different advance ratios and a mesh convergence study. The results of mesh convergence study show that errors for mesh 2 are acceptable (close to 1%) since our main goal is to assess our methodology.

REFERENCES

Caldas, A., Meis, M., Sarasquete, A. (2010) CFD validation of different propeller ducts on Open Water condition. 13th Numerical Towing Tank Symposium, Germany, 2010

STAR CCM + (2009) User Guide (Version 5.02)