Fifth International Symposium on Marine Propulsors
smp'17, Espoo, Finland, June 2017

Numerical Study of Propeller Ventilation

Camille Yvin[1], Pol Muller[1], Kourosh Koushan[2]

[1]DCNS RESEARCH/SIREHNA, Nantes, France
[2]SINTEF Ocean, Trondheim, Norway

ABSTRACT
This paper presents a numerical study of the ventilation phenomenon based on model tests of a pushing thruster with variable advance ratios at a constant immersion depth to radius ratio h/R=1.6. This study uses STAR-CCM+, a commercial CFD code which uses a VOF formulation (Volume Of Fluid) in order to capture the free surface and the air caught by the thruster.

Variations of the methodology are conducted in order to compute propeller thrust and torque and compare them with experimental measurements. The results show that it is still a challenge to numerically evaluate the thrust and torque losses in a partially or highly ventilated regime. However, the ventilation inception can be correctly detected which is one of the most important information from a practical point of view because, in most of the cases, severe ventilation must be totally avoided to prevent important mechanical damages.

Full scale simulations are also carried out in order to investigate the scale effects. Numerical differences are observed between the model and the full scale for advance coefficients below the critical advance coefficient.

Keywords
Ventilation, Thrust lost, RANS simulation, Volume Of Fluid.

1 INTRODUCTION

The ventilation phenomenon, consisting of suction of air from the free surface into the vicinity of a lifting surface, can be responsible of serious damages to mechanical actuators of propellers. In case of a marine propeller, the harsh variations and instabilities of thrust and torque induced by the ventilation lead to strong variations of the engine load and of the shaft rotation speed. In addition this phenomenon produces excitation of the aft of the ship which may result in unexpected vibrations. In particular the ventilation occurs when the distance between the propeller tip and the free surface is small enough or when the propeller load is high enough. Different ventilation inception mechanism and regimes can be found in Kozlowska et al. (2009).

The aim of this study is firstly to determine if a numerical method is able to detect the ventilation inception which is one of the most important information from a practical point of view and secondly to determine if the modifications of the propeller performances (thrust and torque) can be evaluated with an URANS solver.

Numerical simulation of ventilated propellers has already been performed by different authors. Very often, it is shown that the evaluation of the thrust and the torque is a very difficult task as soon as the deformation of the free surface becomes important and complex. In these conditions, the thrust and torque of the propeller are always numerically over-estimated mainly due to an under-estimation of the entrained air below the free surface. Several hypotheses are commonly proposed and are detailed hereafter.

Due to large simulation costs, the number of simulated rotations is often limited to a few tens which it is far below the number of rotations performed during the experiments. Thus, it is possible that some ventilation phenomena do not have enough time to appear (Kozlowska et al. 2011, Wöckner-Klume 2013).

An incompressible flow can also be a too strong approximation, especially in the ventilated area where a mixture of water and air is encountered (Califano 2010). Moreover, the typical VOF approach used to take into account the free surface in simulations is not dedicated to model the air bubbles and the mixture of water and air which can also explain the deviations between experiments and simulations (Kozlowska et al. 2011).

It is also shown that results can be very sensitive to numerical parameters. It has been suggested that the very low pressure inside the tip vortex core seems to play an important role in the ventilation (Califano 2010), but it is well known that it is numerically a very difficult task to predict this pressure accurately (Kozlowska et al. 2011, Yvin & Muller 2016).

2 TEST CASE AND PROPELLER GEOMETRY

Experiments were conducted in the large towing tank of SINTEF Ocean (ex. MARINTEK). General information and experimental setup are available in Koushan et al. (2009).

The propeller is four bladed with a diameter of D=0.25 m (model size). The hub diameter is about 65 mm. The design pitch ratio and the blade area ratio are about 1.1 and 0.595 respectively. The camber line is a NACA a=0.8 line and the profile of the sections is a NACA 66 modified for marine applications. Geometrical quantities are given at the Table 1. A visualisation of the geometry is proposed at Figure 1.

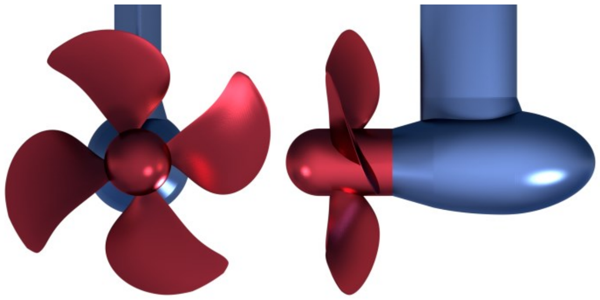


Figure 1: Geometry of the propeller and the pulling thruster

Experiments are conducted for different immersion ratios h/R where R is the propeller radius and h is the vertical distance between the calm free surface and the shaft line. In this study, an immersion ratio h/R=1.6 is considered because the ventilation effect is clearly observable in this case. Open water data are also available. They are obtained for a large immersion ratio which, in the case of the numerical simulations, is set to h/R=3 (h/R=2.5 for experimental tests). All these results are summarized in Figure 2.

Table 1: Sections characteristics

r/R	c/D	t/D	s/D	P/D	f/D
0.24	0.13	0.038	0.000	1.08	0.001
0.26	0.15	0.037	0.003	1.08	0.004
0.30	0.18	0.035	0.011	1.08	0.007
0.37	0.23	0.031	0.023	1.09	0.009
0.46	0.29	0.026	0.037	1.09	0.012
0.57	0.34	0.022	0.045	1.10	0.013
0.67	0.38	0.017	0.040	1.10	0.014
0.78	0.40	0.013	0.014	1.09	0.012
0.87	0.38	0.010	-0.030	1.06	0.010
0.94	0.32	0.008	-0.082	1.00	0.006
0.98	0.21	0.006	-0.125	0.95	0.003
1.00	0.03	0.006	-0.141	0.94	0.000

During experiments, the rotation rate is constant and equal to 14 Hz for every advance coefficient J.

3 NUMERICAL SETTINGS

The numerical towing tank developed by DCNS Research is mostly focused on CFD techniques using finite volumes to solve the incompressible URANS equations. To deal with the free surface problem, a VOF approach is used. Turbulence modelling is achieved with a k-ω SST turbulence model and wall functions are used in the vicinity of the boundary layers.

The computation domain is a parallelepiped with the following dimensions: 4D upstream, on sides and below the propeller, 4D above the calm free surface and 12D downstream. The propeller is inside a cylindrical domain which diameter is about 1.3 larger than the propeller one. The meshes are refined in areas of interest: vicinity of the propeller, wake of the propeller and of the pulling thruster, boundary layers, free surface, leading and trailing edges of the blades as it is shown in Figure 3. The length of the cells on the blades is between $D/2^7$ and $D/2^{11}$. At model scale, when J=0.4, the thickness of the first cell is about 0.4 mm to obtained a y+ close to 70 and 8 cells are used to mesh the boundary layer thickness. A mesh is approximately made up of 9 million of cells.

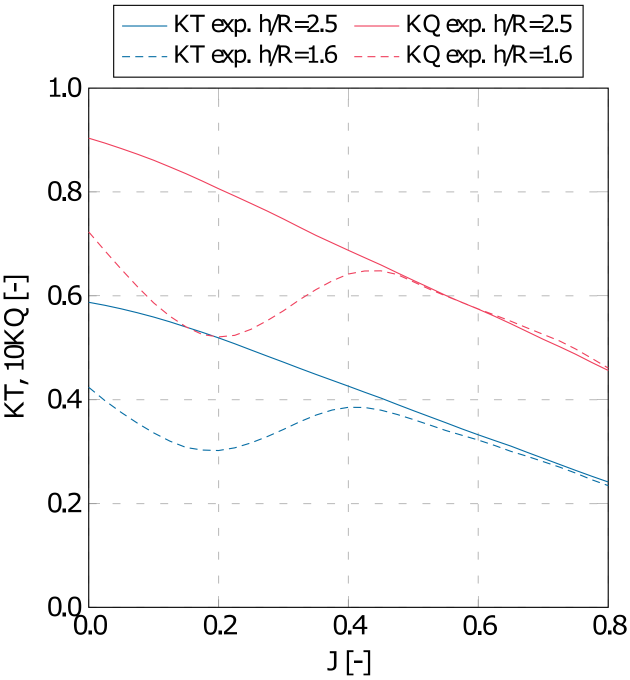


Figure 2: Experimental results

A sliding interface technique is used to take into account the rotation of the propeller. A MRF (Moving Reference Frame) approach was also tried out because it allows larger time steps. Unfortunately, non-physical free surface deformations were observed as soon as the free surface went into the volume where the MRF is applied. That is the reason why the MRF approach was abandoned even if it seems possible to get good results with this approach at bollard condition (Guo et al. 2014).

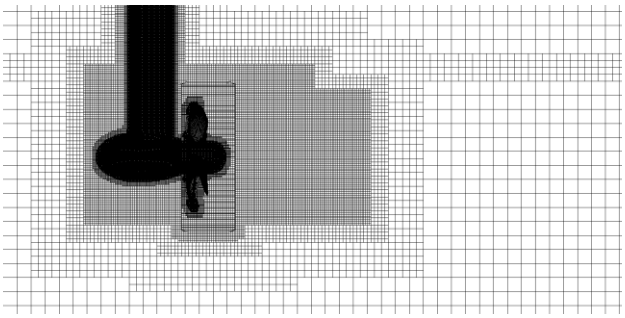


Figure 3: Mesh cut in the vertical and longitudinal plane

The time step is chosen in order that the propeller rotation is about 2° per time step. A time step twice smaller was also used but no significant difference was observed on thrust and torque. The simulation of twenty revolutions requires one day of computation with 80 CPUs.

4 RESULTS

4.1 One simulation per advance coefficient

A first set of simulations was carried out. For each advance coefficient, one different simulation is used. The flow fields are initialized with the inflow conditions and the rotation rate of the propeller is initially set to the final value which is 14Hz. 20 revolutions are simulated. Due to unsteady effects, the hydrodynamic forces are averaged during one revolution for the ten last revolutions (see Figure 9 for instance). This figure also shows that it is difficult to define a representative mean value for the thrust and the torque as soon as the ventilation effects are present. The reader must be aware that the results can be quite different with a different post-processing.

Results are summarized in Figure 4 where the dots and the error bars are the mean values and the min-max respectively. Thrust and torque obtained for h/R=3.0 are also presented at this figure. The numerical forces are in good agreements with the experimental ones when the immersion ratio h/R is large enough to be in "open water" conditions. In that case, deviations of thrust and torque are between -3% and 2%.

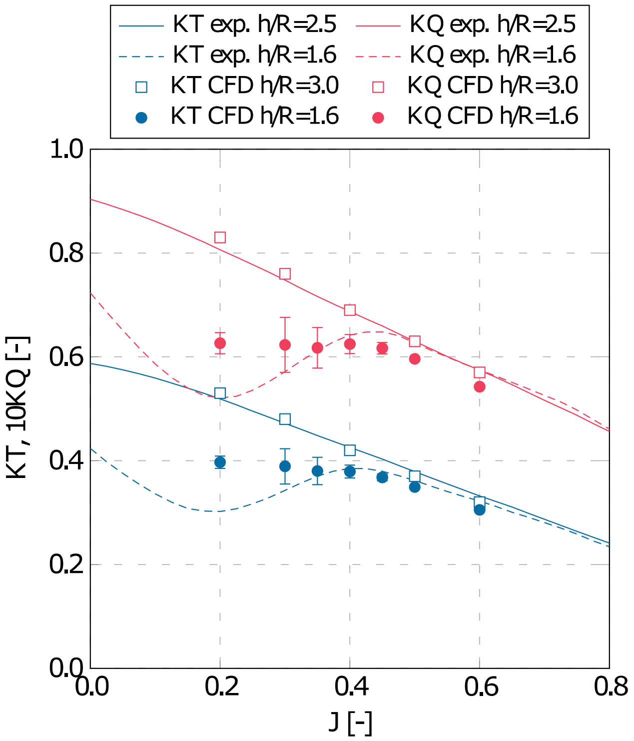


Figure 4: Numerical vs. experimental results - One simulation per advance ratio

For advance coefficients larger or equal to 0.4, the ventilation effects are quite well captured and the thrust estimation is satisfactory. Torque estimation is slightly underestimated. By comparing the deeply immersed cases with the h/R=1.6 cases, it can be said that the effect of the free surface is more emphasized numerically than experimentally. According to Koushan et al (2009), the advance coefficients J=0.4 is the critical advance coefficient for this propeller. For advance coefficient higher than the critical advance coefficient, the ventilation is not entirely developed yet (sub-critical regime). The free surface is modified but the blades are not in contact with the air (see Figure 11.a to Figure 11.c).

For advance coefficients lower than the critical advance coefficient, the thrust and torque are overestimated by simulations. It can be seen in Figure 11.d Figure 11.e that the air starts to be sucked down in the propeller area but the ventilation inception does not occur as it can be expected. Indeed, a partially ventilated regime was expected at least. At this regime, the air is normally captured inside the tip vortex which is not the case here. As it was mentioned in the introduction, this can be explained by different reasons: unsuitable turbulence modelling, simulations run for a too short time, inability for the VOF model to take into account the mixing of air and water, too diffusive numerical scheme to correctly capture the tip vortex, incompressible flow hypothesis, etc. In this study, we decided to try to modify the initial conditions in order to artificially help the ventilation inception because it is a simple thing to do with this chosen numerical model (see section 4.2). This approach was expected to get around the issue of the not enough simulated revolutions and, thus, decrease the simulation costs and access to other ventilation regimes.

In a nutshell, this first set of numerical simulations can be used to detect the very beginning of the ventilated regime which is characterized by an important bending of the KT and KQ curves at this immersion depth. At the opposite, it is unsuitable to correctly evaluate the thrust and torque losses as soon as the ventilation is partially or fully developed.

4.2 One simulation with variable rotation rate

In order to help the inception of the ventilation and start with a fully developed ventilation regime, it was decided to start a simulation with a low advance coefficient (J=0.2) and gradually decrease the rotation rate to get back to the experimental conditions (n=14Hz) for the advance coefficient J=0.3 (see Figure 5) where quite important thrust and torque losses are expected.

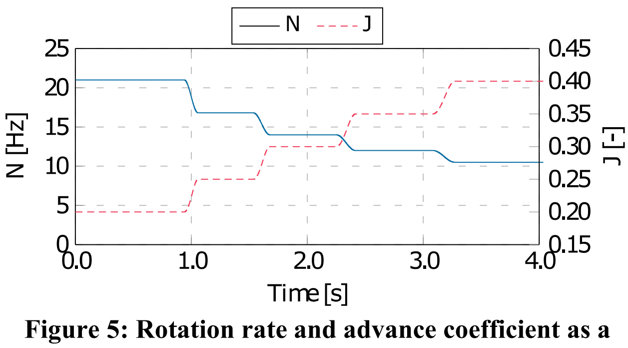


Figure 5: Rotation rate and advance coefficient as a function of time

The inflow velocity remains constant during the simulation. The time step is dynamically modified in order that the rotation of the propeller is about 2° at each time step. Each advance coefficient is simulated during at least 10 revolutions.

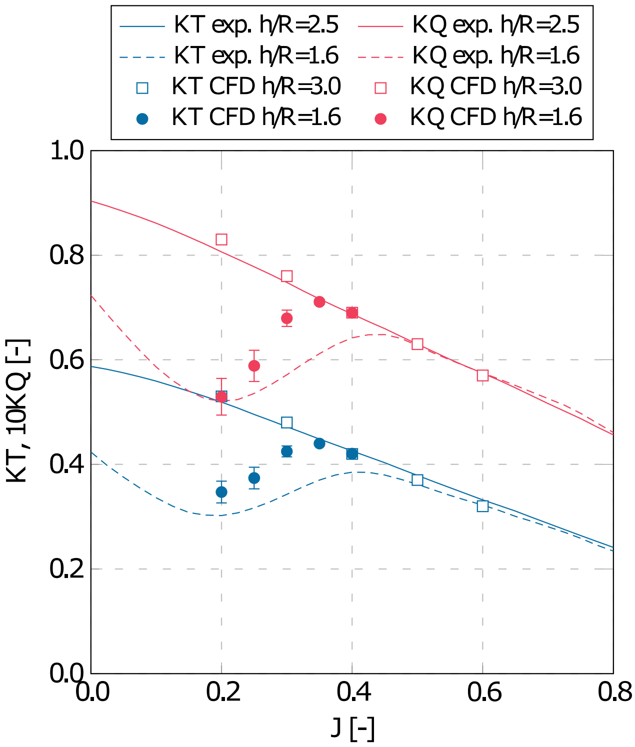


Figure 6: Numerical vs. experimental results - One simulation with variable rotation rate

As expected, the ventilation is far more developed at the beginning of the simulation (J=0.2 and n=21Hz) than during the previous set of simulations (J=0.2 and n=14Hz, compare Figure 11.f with Figure 12.b or Figure 8.a with Figure 8.b). This leads to an important thrust loss (see Figure 6) which is quite close to the measured one but this is just a coincidence because the physical conditions are not the same (same J but higher rotation rate). Unfortunately, even if the flow is highly ventilated at the beginning of the simulation (here the simulation begins with J=0.2), the free surface perturbations are quite identical, as are the forces (see Table 2) when the simulation and experimental conditions are once again the same, namely for J=0.3 (compare Figure 11.e with Figure 12.b or Figure 8.d with Figure 8.e).

Table 2: Thrust and torque for J=0.3

Case	J [-]	KT [-]	10KQ [-]
Exp.	0.3	0.34	0.57
CFD 4.1	0.3	0.39±0.03	0.62±0.05
CFD 4.2	0.3	0.42±0.01	0.68±0.02

The comparison between this simulation and the previous set of simulations for the advance coefficient J=0.2 illustrates also that the kinematic similarity (conservation of the advance coefficient, Kozlowska et al 2009) is not sufficient to scale the results from model scale to full scale even if the geometrical similarity is satisfied because different ventilation regime are observed. Depth Froude's number or ventilation number must also be respected (Kozlowska et al 2009). They are respectively defined as:

$$F_{nh} = \frac{\pi n D}{\sqrt{gh}} \qquad \sigma_V = \frac{2gh}{V_\infty^2} \qquad (1)$$

To summarise, even if the ventilation inception is artificially stimulated, a partially or fully ventilated regime is not obtained with this approach. In other words, initial conditions do not seem to play an important role because identical results are obtained with different initial conditions. This answers to some extent the question about the influence of the initial conditions (Kozlowska et al 2011). However this must be checked with another kind of approach where the velocity is dynamically modified instead of the rotation rate which is what was done during the experiments.

4.3 One simulation with variable towing velocity

To dynamically change the inflow velocity with a VOF model, the approach must be slightly different. Indeed, the inlet velocity cannot be modified during a simulation without producing an incoming wave. In order to solve this issue, the meshes must be dynamically translated at velocity equal to the inflow velocity in the same way as the propeller is towed in an experimental basin.

This simulation was performed but the solution is unphysical (presence of air upstream the propeller) even if it numerically converges well. This seems due to numerical instabilities with the VOF model. Even if the two approaches (inflow velocity or translation of the mesh) are theoretically identical, the solved equations are not the same (inflow velocity is only imposed at the boundary conditions for the first approach whereas it is set to zero and an additional term representative of the mesh velocity appears in the convective part of the equations for the second one). Thus, this approach was given up for now for numerical reasons.

4.4 One simulation per advance coefficient at full scale

Full scale simulations were also performed to investigate the scale effects. The numerical settings are identical to those presented at section 4.1. A discussion about scale effects is proposed in Kozlowska et al (2009).for instance. The scale factor is set to 1/8. The following numbers are preserved: advance ratio, depth Froude's number and ventilation number (see equations 1 for the definition of the two last numbers).

At the opposite, the cavitation number, the Weber's number and the Reynolds' number are not satisfied. The non-respect of the cavitation number and the Weber's number is not a problem here because an analysis of the pressure fields at model scale showed that the water is not supposed to cavitate and the surface tension is numerically not taken into account. Moreover, according to Kozlowska et al (2009), surface tension has no influence here because at both scales the Weber's number is greater than 180 which is the upper bound where the surface tension must be taken into account. Of course it can be an issue to extrapolate experimental results from model scale to full scale but this exercise consists in a comparison between numerical simulations only. Thus, only the Reynolds effects are different here. Meshes are also identical at the exception of the boundary layer which is meshed differently according to the Reynolds' number at 0.75R:

$$Re_{0.75R} = \frac{c_{0.75R}\sqrt{V^2+(0.75\pi nD)^2}}{\nu} \qquad (2)$$

At J=0.3, $Re_{0.75R}$ is equal to7.3 10^5 at model scale (n=14 Hz) and 1.7 10^7 at full scale (n=4.95 Hz). The results at full scale are presented at the Figure 7.

Thrust and torque are quite identical at full scale and model scale (compare Figure 4 with Figure 7) when the advance coefficient is equal or greater than the critical advance coefficient (J=0.4). Means values are quite different below it. The differences between the model scale and the full scale can also be seen on the flow (compare Figure 11 with Figure 13) or the signals of the thrust and torque (compare Figure 9 with Figure 10).

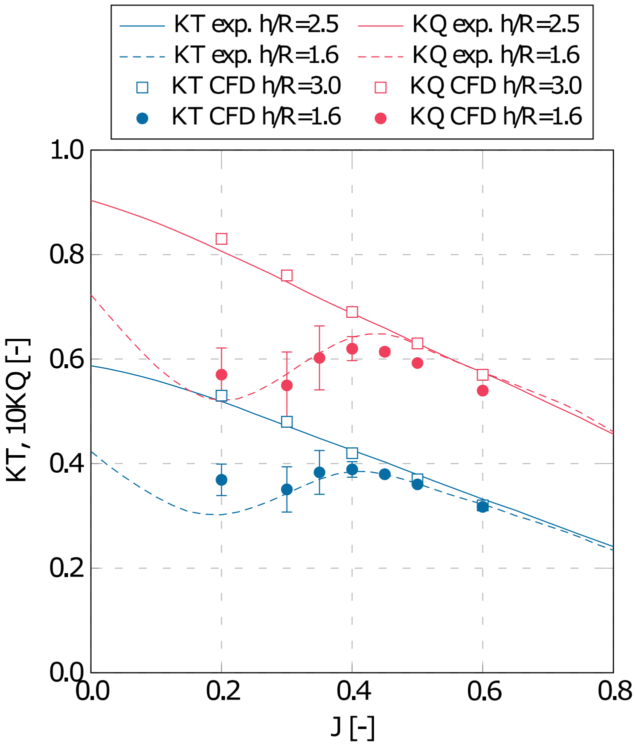


Figure 7: Numerical vs. experimental results - One simulation per advance ratio at full scale

According to these results, it can be said that the Reynolds' number influences the results. Of course, it can be a coincidence if the thrust and torque are better estimated at full scale than at model scale. More simulations are necessary to clearly identify the reason (meshing with y+ below 1 for both scales, decreasing of the time-step…) but it is perhaps a good trail to explain the differences between the experiments and the simulation at model scale (classical turbulence models are not suitable for a Reynolds number below 10^6 which it is the case here).

5 CONCLUSION

This paper presents a numerical study of the ventilation phenomenon. The following conclusions can be drawn:

- Numerical simulations can be used to predict the inception of the ventilation on thrust and torque. This is one of the most important information from a practical point of view to determine the safe operating range and avoid serious mechanical damages.
- It is still very challenging to compute the thrust and torque losses when the ventilation is partially or fully developed. A more complex model seems to be necessary to access the performance for these ventilation regimes. Highly refined meshes, less diffusive model for the turbulence (LES, DES…), numerical method to take into account the mixing of air are possible ways ahead.
- Simulations at full scale show that the Reynolds number seems to affect the mean thrust and torque for advance coefficient smaller than the critical advance coefficient (numerically at least). The too small Reynolds' number at the model can also explain the differences between the experiments and the simulations.

REFERENCES

Califano, A. (2010). 'Dynamic loads on marine propellers due to intermittent ventilation'. PhD Thesis, NTNU, Trondheim, Norway.

Guo, C.-y., Zhao, D.-g. & Sun, Y. (2014). 'Numerical simulation and experimental research on hydrodynamic performance of propeller with varying shaft depths'. China Ocean Engineering, **28**(2), pp 271-282.

Kozlowska, A. M., Steen, S. & Koushan, K. (2009). 'Classification of different type of propeller ventilation and ventilation inception mechanism'. Proceedings of the First International Symposium on Marine Propulsors, SMP'09, Trondheim, Norway.

Kozlowska, A. M., Wöckner, K., Steen, S., Rung, T., Koushan, K. & Spence, S J.B. (2011). 'Numerical and experimental study of propeller ventilation'. Proceedings of the Second International Symposium on Marine Propulsors, SMP'11, Hamburg, Germany.

Koushan, K, Spence, S. J. B. & Hamstad, T. (2009). 'Experimental investigation of the effect of waves and ventilation on thruster loadings'. Proceedings of the First International Symposium on Marine Propulsors, SMP'09, Trondheim, Norway.

Wöckner-Klume, K. (2013). 'Evaluation of the unsteady propeller performance behind ships in wave'. PhD Thesis, Universitätsbibliothek der Technischen Universität, Hamburg-Harburg, Germany.

Yvin, C, & Muller, P. (2016). 'Tip vortex cavitation inception without a cavitation model'. Proceedings of the 19th Numerical Towing Tank Symposium, NUTTS'16, St Pierre d'Oléron, France.

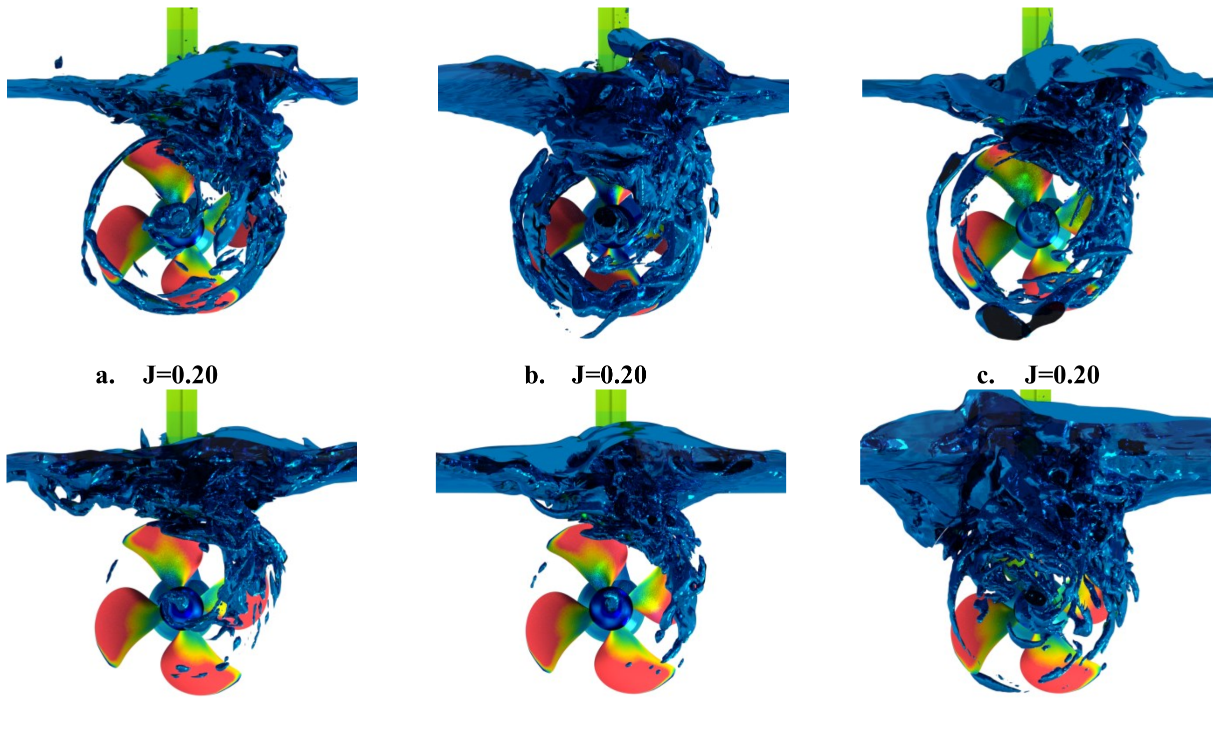
DISCUSSION

Question from Serkan Turkmen

Is numerical model correct at J=0.30? There is wave developed before the strut. Initial conditions, such as inlet velocity, density of the fluid, could be wrong.

Author's closure

Yes, initial conditions and numerical models have been checked. This incoming wave may come from a lack of convergence of the simulation (transient phenomenon), as it is a restart from a previous simulation (J=0.25), see Figure 5 for the kinematic conditions used.



a. J=0.20 **b. J=0.20** **c. J=0.20**

d. J=0.30 **e. J=0.30** **f. J=0.30**

Figure 8: Free surface – Two different advance coefficient – One simulation per advance coefficient at model scale (left), one simulation with variable rotation rate at model scale (middle) and one simulation per advance coefficient at full scale (right)

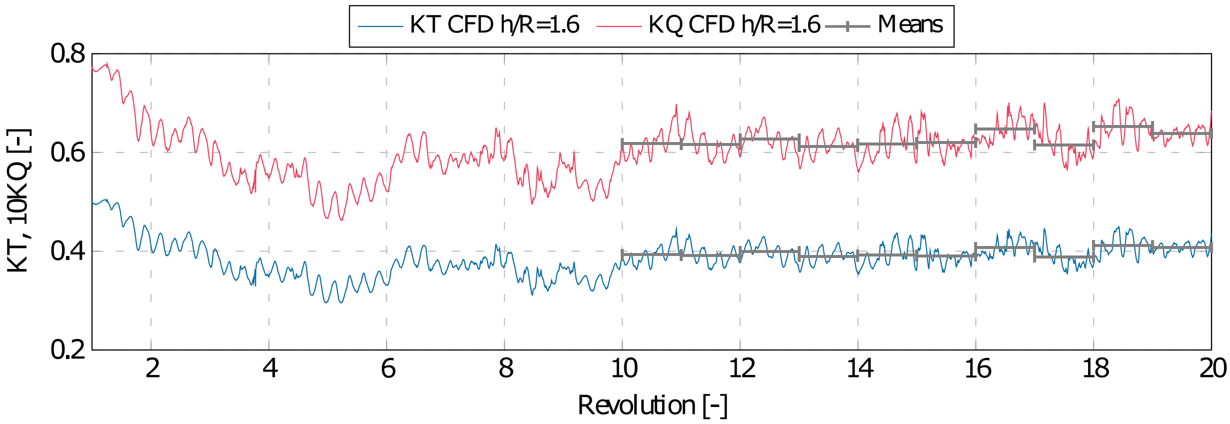


Figure 9: KT and KQ as function of revolution for J=0.2 at model scale

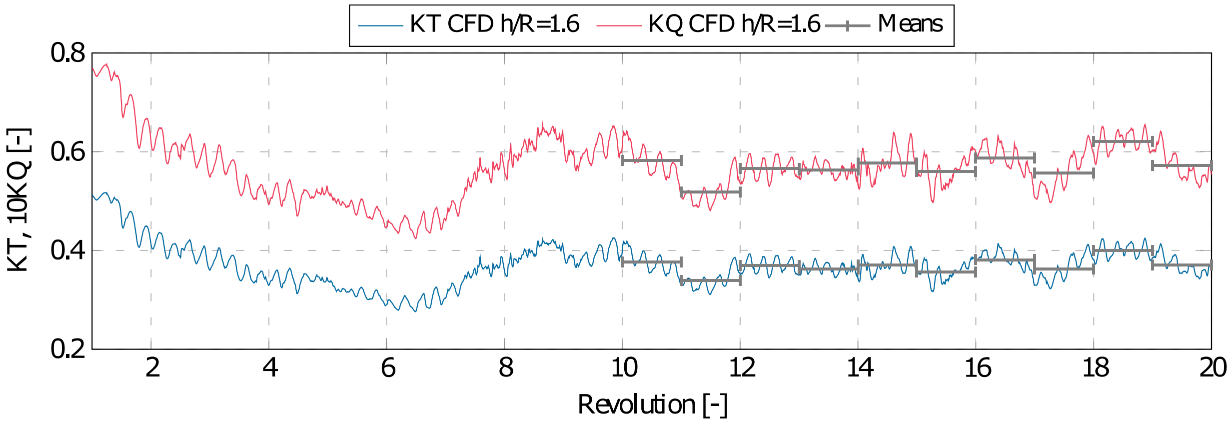
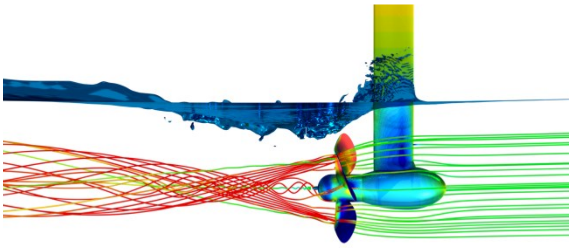
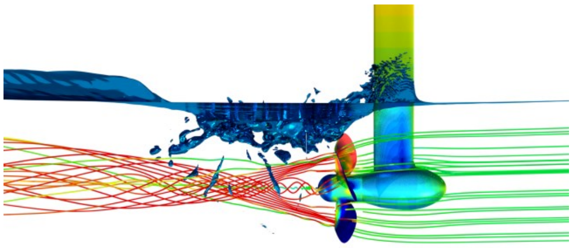


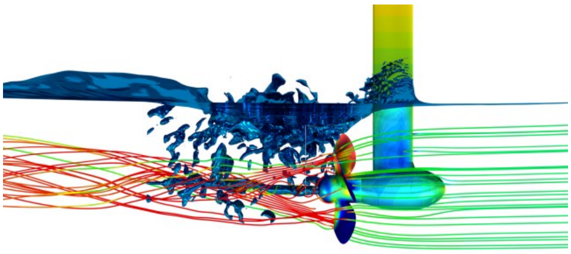
Figure 10: KT and KQ as function of revolution for J=0.2 at full scale



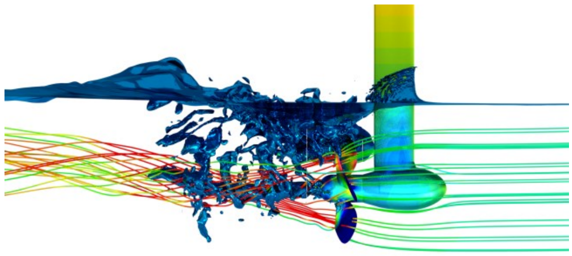
a. J=0.50



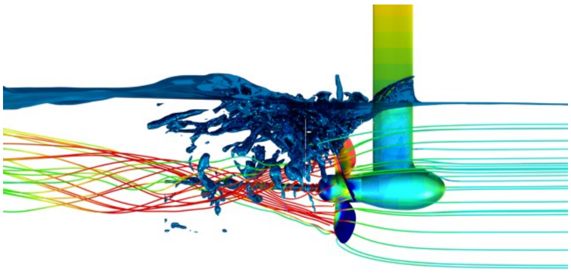
b. J=0.45



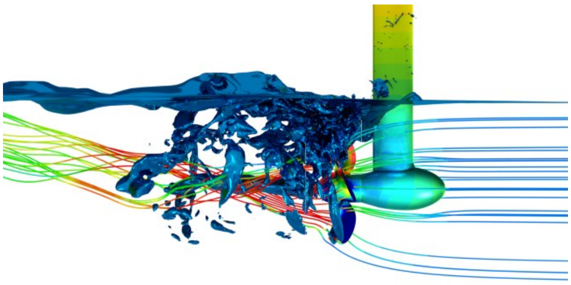
c. J=0.40



d. J=0.35

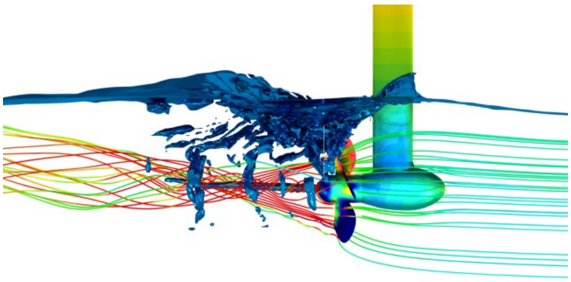


e. J=0.30

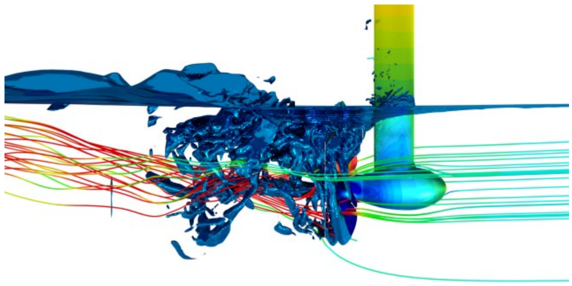


f. J=0.20

Figure 11: Free surface and streamlines - One simulation per advance coefficient

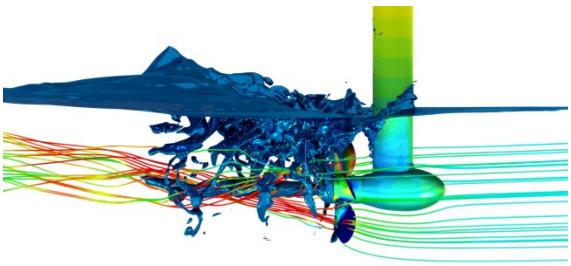


a. J=0.30

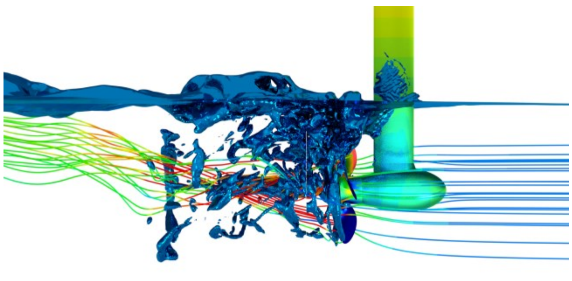


b. J=0.20

Figure 12: Free surface and streamlines – One simulation with variable rotation rate



a. J=0.30



b. J=0.20

Figure 13: Free surface and streamlines – One simulation per advance coefficient at full scale