CFD simulation of a torpedo swim-out launching

F. Pécot (Naval Group / Sirehna), T. Taillefet (Naval Group)

Abstract — Among the various missions expected from a submarine, weapons launching from a tube is of major importance. To secure these launchings and to determine the safe operating envelope, a numerical approach constitutes an interesting alternative to expensive model or full-scale trials. This paper focuses on the development of a numerical methodology based on CFD to simulate the swim out launching of torpedoes from a tube. The methodology relies on the overset grid approach. This tool solves the strongly coupled URANS and 1-DOF weapon dynamics equations, with given propellers rates of rotation. To validate this numerical methodology, full-scale trials of swim out launchings were carried out with a torpedo-like drone from a mono-diameter tube immersed in sea water at rest. A satisfactory correlation was obtained between numerical and experimental results in terms of both weapon velocity and pressures inside the tube

1 Introduction

Among the various missions expected from a submarine, weapons launching from a tube, either by water or mechanical pulse, or in swim out, is of major importance.

At each weapon is associated a safe operating envelope. The success of a weapon launching and of course the safety of the submarine and its crew during this operation is guaranteed provided that the triplet [immersion depth, submarine velocity, sea state] belongs to the firing domain. To determine this domain, the use of a numerical approach able to predict the weapon hydrodynamic behaviour and identify the risks of launching failure constitutes an interesting alternative to expensive model or full scale trials.
In this context, numerical methodologies for the simulation of weapons launching and in particular for the swim out of torpedoes have been developed by Naval Group, on the basis of the CFD code STAR-CCM+ [1]. Hydrodynamic calculations of the swim out firing of a torpedo-like drone from a tube were performed and the obtained results were compared with those of full-scale sea trials achieved in order to qualify the developed numerical tool.

2 Sea trials

2.1. Experimental set-up and measurements

Full-scale trials of swim out launching of a torpedo-like drone from a mono-diameter tube (closed at its bottom) were carried out in sea water at rest, to provide experimental data to validate the proposed numerical CFD methodology. The experiments were performed at an immersion depth of about 11 m on the tube axis, in order to avoid, or, at least, to limit cavitation inception.

The tube length was 6.6 m with an internal diameter of 730 mm (figure 1). Thanks to four rails placed on a 537 mm diameter cylinder, the drone was guided in translation. In the upper rail, a groove had been machined to permit the drone vertical pin to slide and thus avoid any roll motion.

The drone length was 5.8 m (figure 2), with a maximum diameter of 533.4 mm. The slight buoyancy (-14 kg) of the body allowed to get it back easier after shot. The mobile was propelled by a pair of counter-rotating propellers, whose RPM were, for the upstream one, linked to the velocity command (V_{min} = 11 knots or V_{max} = 20 knots) and for the downstream one, determined in order to annul the total torque.

During the trial, a buffer within the drone recorded both the rates of rotation and the acceleration, thus leading to the mobile velocity and displacement. Pressure measurements along and at the bottom of the tube were also achieved, with a 10 kHz sample frequency.

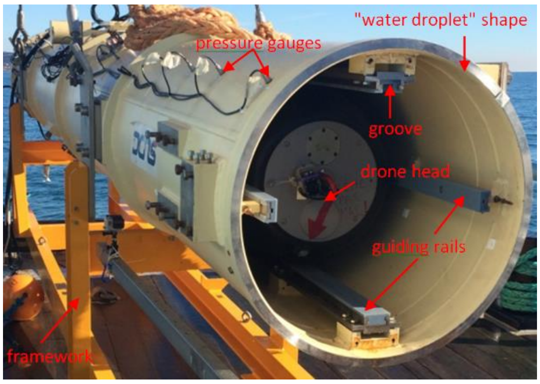


Fig. 1. Torpedo-like drone within the 730 mm diameter launching tube.

Fig. 2. Torpedo-like drone.

2.2. Trials reproducibility and studied configurations

In order to ensure results reproducibility, two trials were performed for each velocity command. Figure 3 and 4 compare the evolutions of, respectively, the propellers RPM versus time, and the drone velocity versus the drone aft position, in both similar experiments, for the V_{max} velocity command. They show that this reproducibility is quite satisfactory.

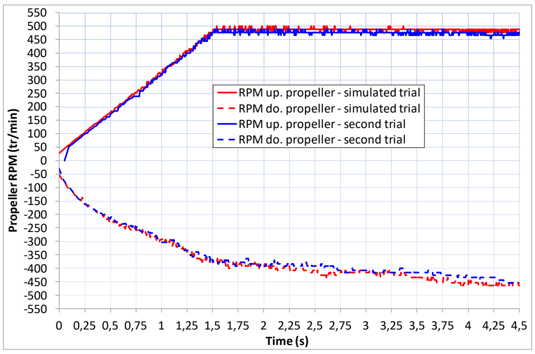


Fig. 3. Trials reproducibility - both propellers RPM - Vmax velocity command.

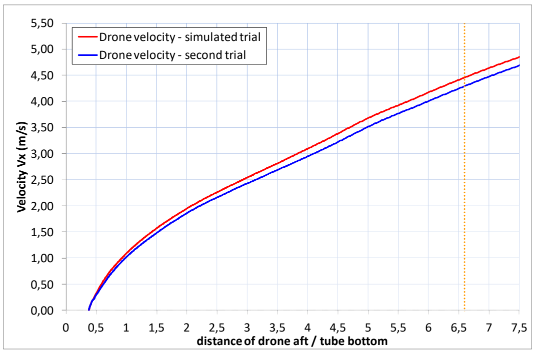


Fig. 4. Trials reproducibility - drone velocity - Vmax velocity command.

3 Numerical CFD approach

With the ongoing computational capabilities and the CFD progress, the swim out launching of a weapon from a tube can now be simulated with a full CFD numerical approach. The selected code must be able both to deal with the time evolution of the calculation domain due to the weapon displacement, and to solve the strongly coupled URANS and 1-DOF weapon dynamics equations, with given propellers rotation rates. In the current study, the ability to solve this problem and the accuracy of the CFD code STAR-CCM+ are evaluated.

3.1. Tube modelling and “overset” approach

Due to the presence of the guiding rails, a very small gap (1.8 mm) is located between the drone and the tube on one hand, and the tube cross section area is reduced on the other hand.

Two different approaches were used to model the tube and its rails, but by strictly keeping the same overall cross section area:

- a simple one [6]: for which this tube was replaced by an equivalent one without any rails, which permits to avoid meshing the small gap,
- a complex one: for which fictive guiding rails were effectively taken into account. The gap between the drone and the modelled rails was chosen as small as possible, but deliberately increased in order to keep enough cells between both bodies. Two distances values between diametrically opposite rails were considered: 560 and 580 mm (Figure 5). Therefore, to keep the same cross section area, the rails were enlarged with respect to reality.

Moreover, the "water droplet" shape at the tube exit was modelled.

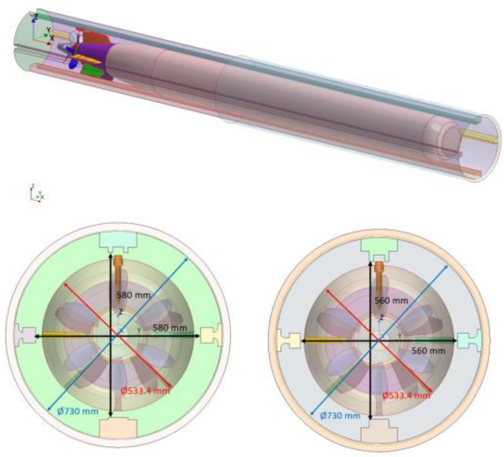


Fig. 5. 730 mm diameter tube + modelled guiding rails (580 and 560 mm distance)

Unlike previous studies on weapons launchings [2, 3, 4], in order to avoid the remeshing of the evolving fluid calculation domain due to the drone displacement, the "overset" grid method available in the STAR-CCM+ was used.

The "overset" method consists, for this current application, in superimposing two non-deforming meshes:

- an overset mesh around the moving drone and its propellers,
- a fixed background mesh inside, around, and outside the tube,

These meshes exchange information data by interpolation between them to solve both the flow and the drone dynamics (Figure 6).

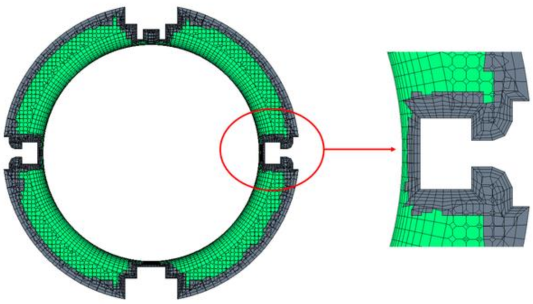
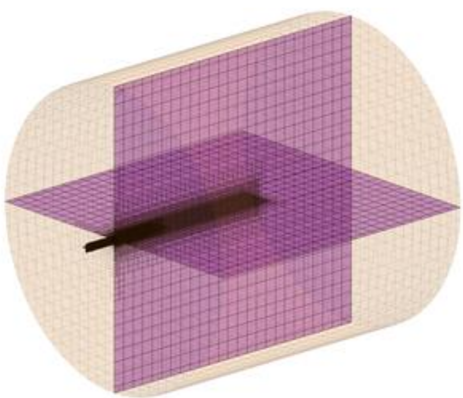


Fig. 6. Meshing of a tube with rails modelling – overset interfaces

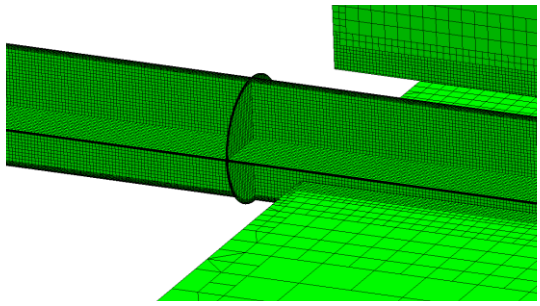
3.2. Meshing

The numerical model included four different regions:

- the background region meshed with trimmed hexahedral cells. The inner launching tube and its exit were homogeneously and finely meshed. But, outside, the mesh was then quickly released (figure 7).
- the overset cylindrical region around the drone and its propellers, which was meshed with polyhedral cells (figure 8). This region was finely meshed close to the drone and inside a cylinder whose diameter is larger than the tube one. Further, the mesh was released.
- two other regions around each one of both propellers (in blue and grey on figure 8), included in the previous overset region.



Overall view of the region



Zoom on the tube

Fig. 7. Views of the background region mesh.

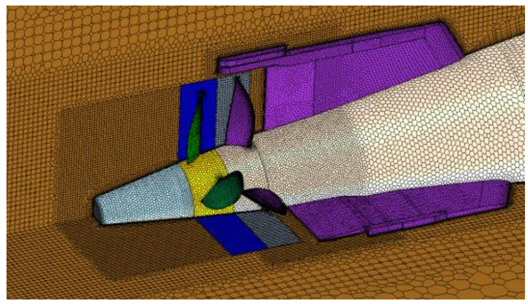


Fig. 8. Surface mesh of the rear part of the drone and its counter-rotating propellers.

3.3. Propellers rotations treatment

To take into account the rotation effects of the counter-rotating propellers on the flow, two methods were compared:

- the "MRF" method (Moving Reference Frame) consisting in solving the Navier Stokes equations in two rotating frames (one for each propeller) without any visible rotation of the propellers,
- the "sliding grid" method consisting in simulating the real rotations (in opposite directions) of the propellers during the launching of the drone. In that case, unlike the previous one, non conformal interfaces are used between the three regions meshes.

For each studied configuration (with or without rails modelling), the global mesh included about 10 millions of cells.

3.4. Modelling

To simulate the drone launchings from the tube with STAR-CCM+ code version 10.06, the following assumptions were taken into account:

- the flow was unsteady, non compressible, turbulent, and in almost all cases monophasic. Nevertheless, some tests were carried out with the Schnerr & Sauer [5] cavitation model enabled.

- the gravity was disabled,
- the rotation rates laws of propellers obtained from the simulated experimental tests were imposed,
- the drone moved forward along the tube axis thanks to its own propellers, without any solid friction on the guiding rails,
- the sea water density was equal to 1026 kg/m^3 and the dynamic viscosity to 1,219 kg.m^{-1}.s^{-1}.

The two equations k-ω SST URANS turbulent model was chosen, since it remains still valid in the vicinity of walls where turbulent Reynolds is low and viscous effects are dominating.

At the boundaries of the calculation domain, we imposed:

- no slip conditions on walls,
- zero relative pressure outlet far from the tube.

Moreover, initially, the fluid was assumed at rest.

All the computations were performed with the STAR-CCM+ segregated solver with implicit linearization of the flow equations, using a first order time scheme and second order space schemes.

4 Comparison between trials and simulations without rails modelling

4.1. Drone displacement, velocity and acceleration

Figures 9 and 10 show the drone velocity versus the drone aft position, for both cases of velocity command (V_{max} and V_{min}). On this figure, experimental and CFD results without cavitation are compared. The two vertical lines indicate the exits of both the drone junction and the drone aft out of the tube.

This figure shows that the drone continuously accelerates inside and outside the tube. Moreover, it shows that, for each velocity command, the numerical results correlate quite well the experimental one. Globally, the "MRF" results are closer than the "sliding grid" ones, even if, up to an aft position of 2.5 m, the latter model leads to a better correlation. But, anyway, "MRF" computations are less time-consuming than the "sliding mesh" ones (15 to 20 hours versus 6-8 days, on 80 cores).

The maximum velocity deviations between numerical and experimental results at the tube exit does not exceed 0.2 m/s, which is of the same order of magnitude of the velocities discrepancy at the tube exit between two similar tests.

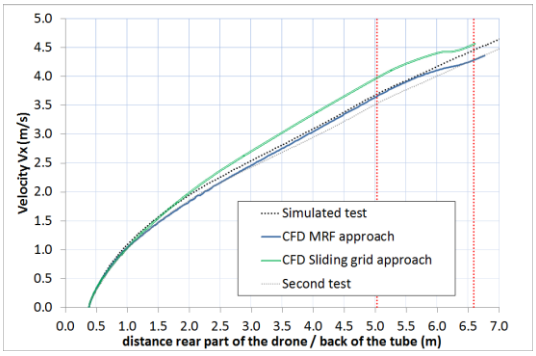


Fig. 9. Drone velocity - V_{max} velocity command – without rails modelling.

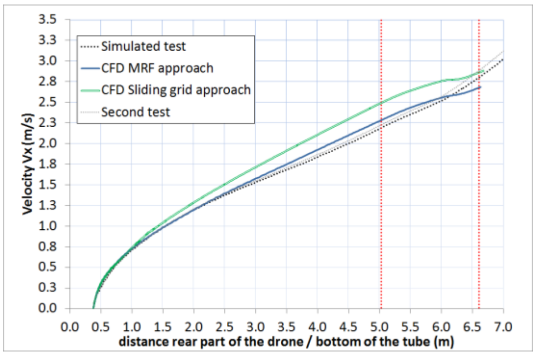


Fig. 10. Drone velocity - V_{min} velocity command – without rails modelling.

In simulations, a significant and sudden drop of the drone acceleration (negative jerk) is observed, while the rear conical part is leaving the tube (figure 11). This acceleration drop, although existing, is far less important during tests and its effect is not visible on the experimental velocities curves. This difference of accelerations behaviours, while the drone rear part is going out, is coherent with the small velocity gap between tests and "MRF" calculations at the tube exit.

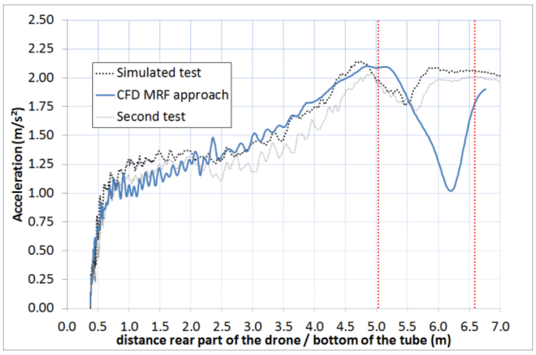


Fig. 11. Drone acceleration –V_{max} command velocity – without rails modelling.

This acceleration drop, generated by a thrust drop, is concomitant with an increase of the average velocity of the incoming flow on the propellers (especially at the blades feet). That is due to the progressive disappearance of a recirculation zone located on the drone upstream of the propellers, during the exit of the conical rear part of the mobile (figure 12).

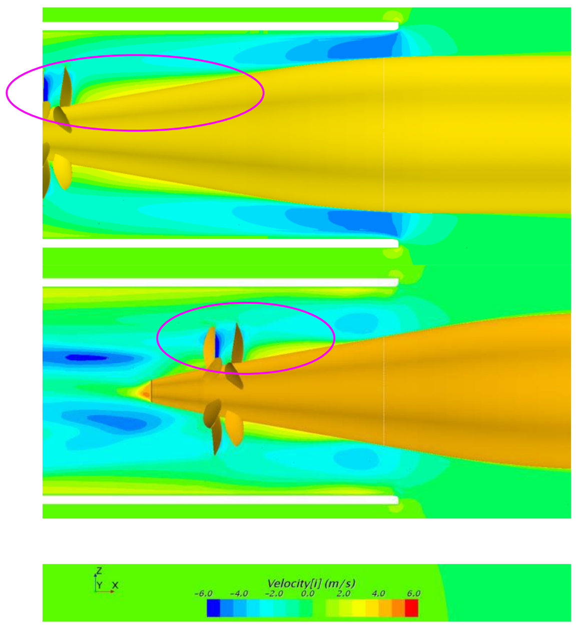


Fig. 12. Disappearance of the recirculation zone upstream of the propellers – without rails modelling (remark: the drone rudders have been voluntary hidden).

To better understand the physics of this phenomenon, a simulation at V_{max} velocity command, without any rudder on the drone was achieved. The obtained results showed that these rudders have no influence on the drone dynamics, including the exit phase of the mobile conical rear part. Therefore, it could be presumed that the tube modelling, in particular the removal of the guiding rails, altering the three dimensional local structure of the flow at the tube exit, is probably responsible for the drone dynamics discrepancies between simulations and experiments.

4.2 Pressure and cavitation inception risk

Figure 13 allows to compare experimental and numerical ("MRF" approach without cavitation) pressure signals versus time, at three different points along the tube, for Vmax velocity command. This figure shows a satisfying correlation obtained at each point between numerical and experimental signals, despite small offsets whose levels depend on the pressure sensors locations.

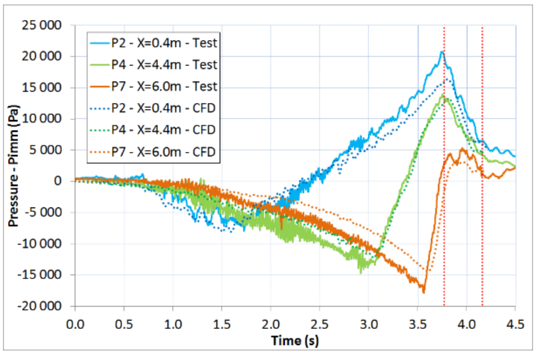


Fig. 13. Relative pressures versus time inside the tube - V_{max} velocity command – without rails modelling.

At each point, the pressure evolves in the same way in three stages:

- a decrease stage, due to the drone motion, making the sea water flow into the tube. Except for P2 point, this phase lasts until the mobile junction overtakes the considered point.
- an increase stage, until the junction goes out, due to the compression of the flow between the drone conical part and the tube (diffuser-like annular space) and at the bottom of this one. It can be noticed that, at this time, the pressure is higher than outside.
- a decrease stage, during which the pressure reaches the immersion one.

Although the cavitation was not simulated, the risk of cavitation inception was, nevertheless, assessed by comparing pressure levels on the blades with the satured vapor pressure value. If the pressure drops below this threshold value, this phenomenon should occur.

According to the "MRF" simulations results, there is no risk of cavitation inception in both cases of velocity command. But, according to the "sliding grid" ones, there are some small potential areas of cavitation inception on the leading edges of the blades extrados (in dark blue on figure 14) in the case of V_{max} velocity command.

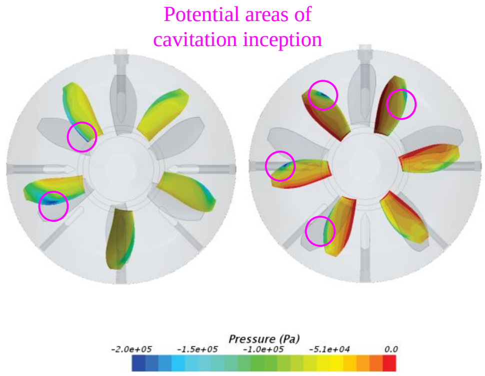


Fig. 14. Cavitation inception risk - "sliding grid" approach - V_{max} velocity command – without rails modelling.

In order to understand, in that case, the impact of cavitation inception on the drone dynamics, an additional "sliding grid" simulation was carried out, with the Schnerr & Sauer [2] cavitation model, whose parameters were kept to their default values. This simulation showed that cavitation actually appears on the blades (figure 15). Nevertheless, there was practically no impact on the dynamics (figure 16).

It can be noticed that this simulation was a bit time-consuming, since it lasted more than 20 days on 80 cores.

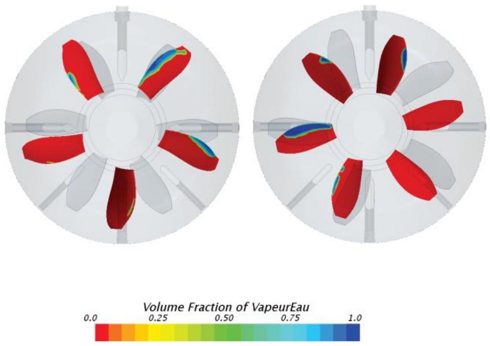


Fig. 15. Vapour volume fraction - "sliding grid" approach - V_{max} velocity command – without rails modelling.

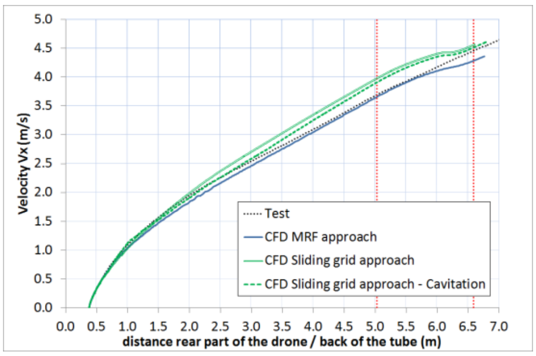


Fig. 16. Drone velocity (with and without cavitation modelling) - V_{max} velocity command – without rails modelling.

5 Comparison between trials and simulations with rails modelling

The impact of the guiding rails modelling on the drone velocity was studied to attempt to explain the observed drone dynamics differences between tests and simulations during the exit of the rear conical part of the body.

Simulations with disabled cavitation model were performed, for V_{max} velocity command, using "MRF" approach in both guiding rails modelling configurations, and also using "sliding grid" approach in the case of 560 mm distance.

5.1. Drone displacement, velocity and acceleration

Figure 17 (zoom on the rear cone part exit phase) allows to compare the evolutions of the drone velocities obtained experimentally and numerically with and without rails modelling, versus drone aft position.

A satisfactory overall agreement is observed between trial and "MRF" simulation results with or without rails modelling. On the other hand, the "sliding grid" simulations systematically overestimate the drone velocity, even if, taking into account guiding rails, improves the correlation.

Figure 18 shows that modelling guiding rails significantly reduces the acceleration drop concomitant to the exit of the drone rear conical part, and all the more the distance between diametrically opposite rails decreases. If this distance was enough reduced to be close to its real value, the simulated acceleration profile should logically fit the trial one, during that phase.

The simulated acceleration drop is related to a thrust drop (figure 19). It is due to an increase of the mean axial velocity (in absolute value) of the incoming flow on the propellers, following the disappearance of a recirculation zone located just upstream of the propellers. This velocity rise is lower when rails are modelled and the distance between diametrically opposite rails is smaller (figure 20). Moreover, the radial distribution of the axial flow velocity in a cross section located just upstream of the propellers is drastically modified by the presence of rails (figure 21).

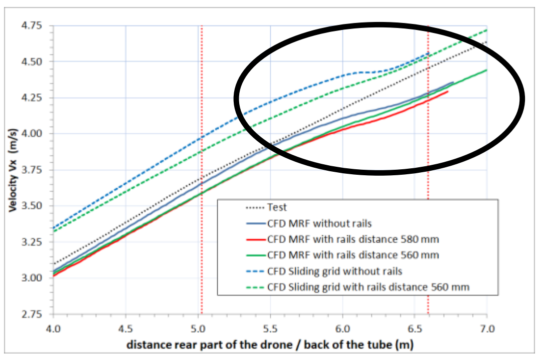


Fig. 17. Zoom - drone velocity with and without rails modelling- V_{max} velocity command

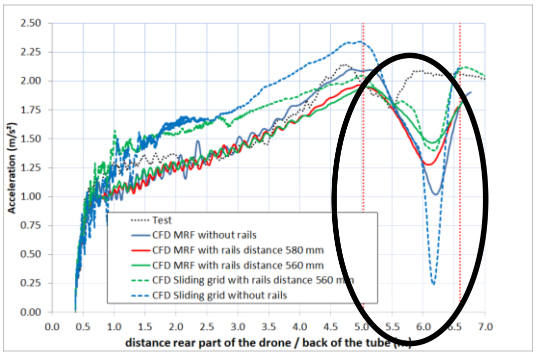


Fig. 18. Drone acceleration with and without rails modelling - V_{max} velocity command

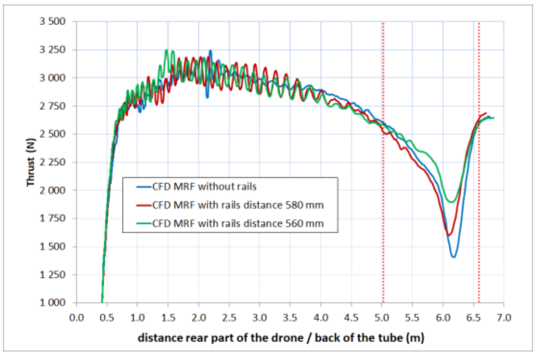


Fig. 19. Propeller thrust with and without rails modelling - V_{max} velocity command

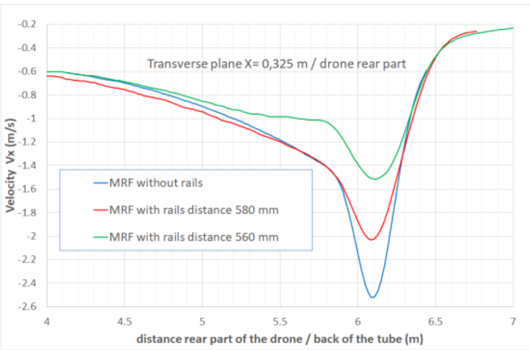


Fig. 20. Mean axial velocity in a cross section upstream of the propellers with and without rails modelling - V_{max} velocity command

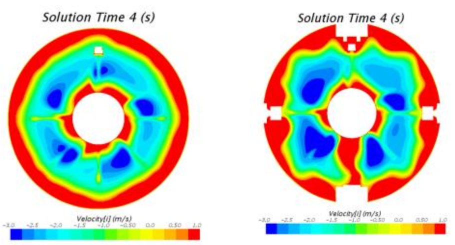


Fig. 21. Axial flow velocity (m/s) upstream of the propellers with and without rails modelling - V_{max} velocity command

5.2. Pressure

Figure 29 allows to compare the static pressures coming from trials and “MRF” simulations with and without rails modelling versus time, at two different positions along the tube.

Guiding rails modelling allowed to further improve the calculation/test correlation in terms of pressures within the launching tube during the exit of the drone rear conical part (figure 22), all the more the distance between diametrically opposite rails decreases.

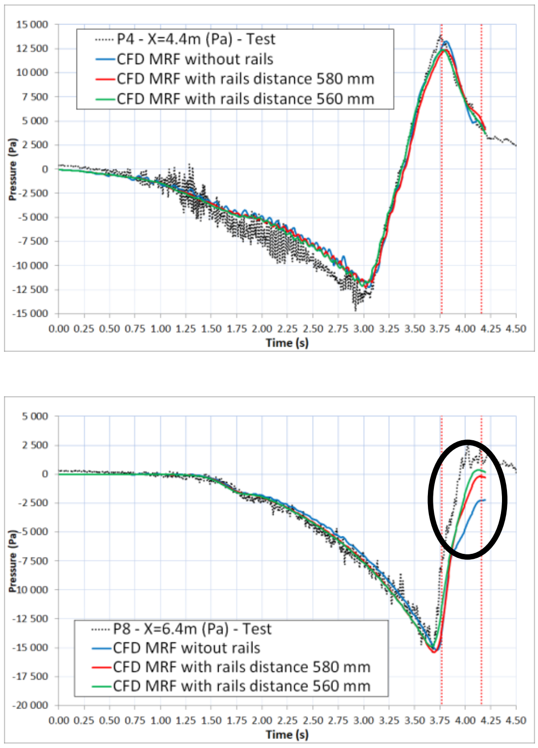


Fig. 22. Relative pressures inside the tube with and without rails modelling - V_{max} velocity command

6 Conclusions

Experimental and numerical results (drone displacements, velocities and accelerations, and pressures inside the tube) generally show satisfactory correlations for the two considered velocity commands, whatever the tube modelling (with or without rails), the used approach to take into account the propellers rotation ("MRF" or "sliding grid"), and the flow modelling (cavitation model enabled or not).

Therefore, the developed methodology on the basis of the STAR-CCM+ code can now be used to predict the torpedoes swim out launchings performances from submarine tubes.

If the real tube with guiding rails is replaced in the model by an equivalent one with the same cross section area, the drone dynamics is not perfectly captured when its rear conical part is leaving the tube. Indeed, the simulations systematically predict an acceleration drop that is overestimated.

Simulations with guiding rails modelling have improved correlations with trials results during the critical phase corresponding to the drone rear cone part exit, since they more accurately describe the local flow just upstream of the propellers.

Nevertheless, the modelling of guiding rails has the double drawbacks of making modelling more complex (cleaning and simplifying CAD, use of finer meshes close to the rails...) and increasing calculations times (since finer meshes contain many more cells), and even making the "overset" method less robust (more complex interfaces between meshed regions).

The best compromise between calculation time and results accuracy is obtained when the "MRF" approach is used with neither rails modelling nor cavitation model activation. If the calculation results really show large areas on the propeller blades extrados where the pressure is lower than the saturated vapour one, a second simulation with enabled cavitation model will be able to be performed.

References

[1] STAR-CCM+, is a CFD solver edited by CD-adapco, http://www.cd-adapco.com

[2] R. Ramaturi, W. Sandberg, R. Löhner, "Simulation of a Torpedo Launch Using A 3-D Incompressible Finite Element Solver and Adpatative Remeshing", NRL/MR/6410--95-7797, (1995)

[3] J.M. Bolot, T. Chambenois, J.M. Laurens, L. Le Saint, X. Merlin, P. Perdon, J.M. Quenez, C. Rigoux, "Numerical pre-qulification weapons launching from a Scorpène submarine", UDT Pacific, (2006)

[4] A. Georges-Picot, B. Pengam, R. Luquet, P. Perdon, "Numerical simulations of interacting multiple bodies launched from a submarine", UDT (2016)

[5] W. Yuan, J. Sauer, G. H. Schnerr, "Modeling and computation of unsteady cavitation flows in injection nozzles", Med. Ind., 383-394, (2001)

[6] F. Pécot & T. Taillefet, "Method for the simulation of the swim-out of a weapon from a launching tube", Warship, 14-15 June, Bath, UK, (2017)

Author/Speaker Biographies

Fabian PECOT has been holding the current position of R&D engineer at Naval Group/Sirehna, for more than 18 years. He is in charge of hydrodynamic technical studies and research, particularly in the field of hydro-mechanical problems

Thierry TAILLEFET has a Ph. D in combustion and has been holding the current position of hydrodynamic expert at Naval Group for 8 years. He has more than 20 years of experience in many fields of fluid mechanics and heat transfer.