Numerical Study of Propeller Diameter Effects for a Self-Propelled Conventional Submarine

Daniel Norrison¹, Kristian Petterson², William Sidebottom¹

¹Defence Science and Technology Group (DST Group), Melbourne, Australia
²Swedish Defence Research Agency (FOI), Stockholm, Sweden

ABSTRACT

Submarines are designed for low drag, manoeuvrability, stability and stealth. As requirements become ever more stringent with each new generation, a better understanding of the relevant fluid dynamic characteristics of such vessels is of increasing importance. This paper builds on previous computational studies by the authors and investigates the effect of changing the propeller diameter on various flow variables. Specifically, a near-wall modelled LES approach was used to simulate the flow around a full-scale, fully-appended Joubert generic submarine. Simulations were carried out in the straight-ahead and self-propulsion conditions at 4.6 kt, resulting in a free-stream Reynolds number of \( Re = 165 \cdot 10^6 \). Two propellers have been considered: a 7-bladed DSTG 057-1 propeller of 5.0 m diameter rotating at 23 rpm and the same propeller scaled to 4.0 m diameter rotating at 33 rpm. Quantitative and qualitative analysis of the propeller surface loads and wake flows indicates that the increased rpm of the 4.0 m propeller leads to higher blade loadings and stronger hub and tip vortices, while the load distribution remains similar to the 5.0 m design. Downstream propeller wake breakdown also occurs much earlier for the smaller diameter propeller.

Keywords

Submarine, LES, propeller, wall-model, turbulence.

1 INTRODUCTION

Operational requirements for submarines are driving a demand for increasingly quiet, more stealthy designs. In turn, successful management and reduction of a submarine’s hydrodynamic signature requires improved and more reliable estimates of the flow physics around the hull and propeller. These flow physics are complex and include a turbulent boundary layer on the hull and appendages; wakes and/or trailing vortices that are generated by the appendages; and a highly turbulent inflow to the propeller that results from the upstream boundary layers and wakes. To control or influence the behaviour and interactions of these features, a greater understanding of their formation is required.

Physical experiments can offer valuable insights into the aforementioned flow features by providing both flow visualisation and quantitative data. Although necessary in the study of hydrodynamics, experimental methods are subject to several limitations. In model-scale experiments, scale effects can be significant. This is particularly true if the model exhibits large regions of laminar flow, resulting in the need for a boundary layer trip. Even when applied correctly, the resultant transition locus when using boundary layer trips rarely matches that of a full-scale boat where a number of transition mechanisms may act simultaneously. Other Reynolds-number scaling effects are even harder to quantify given that scale-models are typically around one fiftieth the size of a full-scale boat. The quality of data resulting from experimental methods can also be adversely affected by poor flow quality and interference from supports used to hold a model. Finally, whilst information on real physical processes can be gathered, the quality and quantity of the resulting data is wholly reliant on the acquisition systems and methods used, which in turn are dependent on budget, model size, mounting method, test configuration and other facility-related constraints.

Computational techniques provide an alternative method for obtaining insight to the flow physics and hydrodynamic performance of a submarine. Computational fluid dynamics (CFD) methods are able to overcome several of the disadvantages associated with physical experiments. CFD allows for calculation of flow variables everywhere in a given domain, non-intrusive data collection, the elimination of support interference effects, and the ability to simulate both model-scale and full-scale geometries. Nevertheless, CFD is also subject to several limitations, many of which are imposed due to limited computational resources and the requirement for validation studies. In order to generate solutions in a reasonable time, simplifications are often necessary. These include reducing geometric complexity, simplifying the flow physics, or a combination of the two, all of which may lead to inaccurate solutions. However, with today’s computational infrastructure and advanced CFD models, it is becoming feasible to perform high fidelity simulations of fully-appended configurations at full-scale (Liefvendahl et al., 2016).

This study builds upon earlier work by Anderson et al. (2012), Fureby et al. (2014) and Norrison et al. (2016) and aims to assess the effect of varying propeller diameter on the steady and unsteady flow physics of a fully-appended conventional submarine at full-scale and in straight-ahead self-propulsion conditions. The investigations are limited to deeply-submerged, non-cavitating flow at 4.6 knots, giving a free-stream Reynolds number based on the hull length and boat speed of \( Re = 168 \cdot 10^6 \). As a baseline case, the
generic 5.0 m diameter 7-bladed DSTG 057-1 propeller design was mated to the DSTG Joubert generic submarine model (Joubert, 2004 & 2006; Loid & Bystrom, 1983; Norrison et al., 2016). A second propeller/hull combination was then produced using a scaled 4.0 m version of the baseline propeller. The investigation is constrained to a numerical study, using Large Eddy Simulations (LES) to provide insights into the flow physics and a method based on the Reynolds-Averaged Navier–Stokes (RANS) equations to compute the self-propulsion performance.

2 GEOMETRY

The DSTG Joubert generic submarine model provides a useful representation of a conventional submarine that can be studied and discussed in the open literature. The geometry stems from the work of Joubert (2004, 2006) and Loid & Bystrom (1983), who proposed a design based on principles consistent with achieving low resistance and flow noise, particularly over the passive bow sonar.

As reported by Norrison et al. (2016), the submarine stern cone has been truncated to accommodate a propeller. The axial position of the truncation is 69.0 m, producing an opening of 1.04 m. This matches the hub diameter of the original 5.0 m diameter DSTG 057-1 propeller. The DSTG 057-1 is a right-handed, generic 7-bladed submarine propeller based on the geometry in Huang et al. (1976) and blade sections in Brockett (1966). The propeller is shown in Figure 1. For further details of the propeller design, see Norrison et al. (2016). When mounted on the submarine, the centre-plane of the propeller is situated at 69.5 m, or about 0.99 L_{oa} (where L_{oa} is the overall length of the boat without the propeller). An elliptical fairing is added to the hub, resulting in a new boat length, L, of 71.2 m, which increases the Reynolds number by 1.42% compared to the boat without a propeller.

The 4.0 m diameter propeller was generated from the baseline 5.0 m diameter DSTG 057-1 propeller by scaling the blades to 80% in all three of their original dimensions using the centreline of the propeller plane as an origin. Fillets were then applied to the intersections of the re-sized blades and the original hub, which remained unaltered. The near-root section of each blade was, however, lost during this scaling process. It should be noted that this approach is not the typical method for producing a scaled-down version of a larger propeller and was adopted here for simplicity.

3 COMPUTATIONAL APPROACH

For a submerged submarine operating at sufficient depth away from any surface effects, the hydrodynamics can be computationally modelled using the incompressible Navier–Stokes equations,

\[ \frac{\partial \mathbf{v}}{\partial t} + \nabla \cdot (\mathbf{v} \otimes \mathbf{v}) = -\nabla p + \nabla \cdot \mathbf{S}, \]

\[ \nabla \cdot \mathbf{v} = 0, \]  

where \( \mathbf{v} \) is the velocity, \( p \) the pressure, \( \mathbf{S} = 2\nu \mathbf{D} \) the viscous stress tensor, \( \nu \) the viscosity and \( \mathbf{D} = \frac{1}{2} (\nabla \mathbf{v} + (\nabla \mathbf{v})^T) \) the rate-of-strain tensor, and \( \nabla \) is the gradient operator. The subscript \( D \) denotes the deviatoric part. Most turbulent flows consist of interacting vortex filaments and sheets, with scales on the order of the Taylor and Kolmogorov scales, \( \propto Re^{-0.5} \) and \( \propto Re^{-0.75} \), respectively, these being orders of magnitude smaller than the integral scales. Clearly, resolving all flow scales by means of Direct Numerical Simulation (DNS) is currently far too computationally expensive for full-scale submarine hydrodynamic problems. Hence, for this study, as with earlier work (Anderson et al., 2012; Fureby et al., 2014; Norrison et al., 2016), RANS and LES methods were used.

3.1 Numerical Methods and Solution Algorithms

In this study, the RANS and LES equations were solved using OpenFOAM, which is based on an unstructured collocated finite-volume method using Gauss theorem together with a multi-step time-integration method (Weller et al., 1997). For LES, the time-integration is performed using a semi-implicit, second-order, two-point backward-differencing scheme. Convective fluxes are reconstructed using multi-dimensional, cell-limited, linear interpolation, whereas diffusive fluxes are reconstructed using a combination of central-difference approximations and gradient-face interpolation to minimise the non-orthogonality error. The Pressure Implicit with Splitting of Operators (PISO) method is used to discretise the pressure-velocity coupling. The scheme is second-order accurate in space and time, and the equations are solved sequentially, with iteration over the explicit source terms and a Courant number of approximately 0.4. The sub-grid scale model used was the Mixed Model (see Norrison et al. (2016) for details). For RANS, a steady-state solver is used together with the Semi-Implicit Method for Pressure Linked Equations (SIMPLE). Convective fluxes are reconstructed using multi-dimensional, cell-limited, upwind-biased interpolation, whereas diffusive fluxes are reconstructed using a combination of central-difference approximations and gradient-face interpolation to minimise the non-orthogonality error.

Figure 1: DSTG 057-1 propeller (Norrison et al., 2016).
The computational methodology used in this study has been used extensively to examine a wide range of flows. Details of a number of cases used for verification and validation of RANS, DES and LES can be found in Norrison et al. (2016).

3.2 Computational Configurations

The computational domain, see Figure 2, is comprised of a cylindrical domain of radius $1.5L_{oa}$ and length $6L_{oa}$. The foremost part of the bow was positioned at the axis origin $(0, 0, 0)$, $1.5L_{oa}$ downstream of the inlet boundary. All grids were generated using a patch-based unstructured Delaunay approach supplemented by between 10 and 15 layers of boundary-layer prisms, see Figure 3. The unstructured mesh was expanded at a rate of 1.07 away from all surfaces, the same expansion ratio being used to control cell growth away from the various density regions necessary to adequately capture the wake flow. To accommodate the propeller, the main mesh was generated with an internal mesh boundary into which a sub-mesh containing the propeller and hub could be inserted. Summary statistics for the hull and the propeller sub-meshes are shown in Tables 1 and 2.

Table 1. Mesh statistics, (all figures in million cells).

<table>
<thead>
<tr>
<th>Mesh Component</th>
<th>Prisms</th>
<th>Tetra</th>
<th>Total</th>
</tr>
</thead>
<tbody>
<tr>
<td>Hull only</td>
<td>27.1</td>
<td>195.9</td>
<td>203.0</td>
</tr>
<tr>
<td>4.0 m propeller</td>
<td>27.9</td>
<td>67.9</td>
<td>95.8</td>
</tr>
<tr>
<td>5.0 m propeller</td>
<td>31.5</td>
<td>74.5</td>
<td>106.0</td>
</tr>
</tbody>
</table>

The boundary conditions for both simulations presented in this paper included constant velocity and zero pressure gradient at the inlet; zero velocity gradient and constant pressure at the outlet; a slip condition on the cylindrical far-field boundaries; and a no-slip condition at the walls with wall functions for the turbulence variables. The internal boundary between the main and sub-meshes was set as an Arbitrary Mesh Interface (AMI) to facilitate propeller rotation. For each case, the flow field was initialised by mapping a previous solution obtained for the submarine without a propeller (Fureby et al., 2014).

3.3 Self-Propulsion Approach

Characterisation of the self-propulsion performance for both propellers was undertaken using RANS with the k-ω-SST turbulence model. The propeller was incorporated into the simulations using a multiple-reference-frame approach coupled with an AMI. To avoid creating two additional meshes, the RANS simulations were computed using the high-resolution LES meshes. At a given boat speed, $U_\infty$, the propeller rotational rate, $n$, for self-propulsion was computed in an iterative manner in order to target a net force of zero for the boat. Since the function of net force versus rotational speed resembles a cubic polynomial with an inflection point at the self-propulsion location, the solution may be obtained using root-finding methods. However, suitable root-finding methods are limited to those which solely use function evaluations since the geometry-dependent function itself is not known a priori for a given $U_\infty$. Thus the Secant method was employed for this study,

$$n_{a+1} = n_a - \frac{n_a - n_{a-1}}{f(n_a) - f(n_{a-1})}$$

where $n_1$ and $n_0$ represent two initial guesses for the rotational rate; and $f(n)$ represents the net force computed by one RANS simulation for a set $U_\infty$. The search for $n$ was continued to a tolerance of $|n_{a+1} - n_a| \leq 0.1$ rpm.

4 RESULTS

4.1 Self-Propulsion Performance

Results from the self-propulsion calculations are presented in Figure 4. As expected, reducing the propeller diameter caused the rpm to increase across the calculated speed range. The magnitude of the increase was typically around 36%.

Figure 3. Schematic of the unstructured computational grid around the fully-appended hull with the 4.0 m-scaled 7-bladed DSTG 057-1 propeller (centre); a zoomed view of the mesh near the trailing edge of the fin (left); and a zoomed view of the mesh around the stern and propeller (right). Some characteristic dimensions are also shown.
Table 2. Summary of computed simulations.

<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>4.0 m DSTG057-1</td>
<td>4.6</td>
<td>423</td>
<td>103</td>
<td>2.20</td>
<td>9.08 rotations (0.55 FPT)</td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>5.0 m DSTG057-1</td>
<td>4.6</td>
<td>23</td>
<td>309</td>
<td>1.01</td>
<td>10.61 rotations (0.92 FPT)</td>
<td></td>
</tr>
</tbody>
</table>

Figure 4: Self-propulsion curves for the Joubert submarine with 4.0 m and 5.0 m diameter propellers.

A quartic polynomial is fitted to the data in Figure 4 to assist with determining the appropriate n needed for the subsequent LES investigations at 4.6 knots. The interpolated n was 24 rpm for the 5.0 m diameter propeller but due to a looser tolerance in initial calculations, a value of 23 rpm was adopted; whilst the value for the 4.0 m diameter propeller was 33 rpm.

4.2 General Flow Characteristics

In order to illustrate the general flow features around the hull, Figure 5 presents a three-dimensional visualisation of \( \lambda_2 = -0.05 \) iso-surfaces. \( \lambda_2 \) is a vortex identification method based on local pressure minima (Jeong & Hussain, 1995). Structures present include horse-shoe vortex systems around the base of the fin and rudders, primary and secondary pairs of counter-rotating vortices at the fin and rudder tips, unsteady fin and rudder wakes, tip vortices shed from the propeller blades and a central vortex extending downstream of the hub (a detailed explanation of these features and their interaction with the boundary-layer flow is given in Norrison et al., 2016). However, the most prominent differences between the 4.0 m and 5.0 m propeller diameter cases are present at the stern as highlighted in Figures 5(b) and 5(c). In particular, the behaviour of the propeller tip vortices which break down more rapidly in the 4.0 m diameter configuration relative to the 5.0 m configuration. The difference in slipstream between the two configurations also appears to change the character of the secondary tip vortices emanating from the upper rudders. However, flow structures produced by the lower rudders are comparable, as are the flow fields over the tapered part of the stern and in between the rudders.

The axial velocity distribution on the meridian plane around the fully-appended hull is presented in Figure 6. Flow behaviour around the bow and mid-section for the 4.0 m and 5.0 m diameter propellers is shown in Figures 6(a) and 6(c), respectively, and appears to be negligibly affected by changes in propeller diameter. As shown in Figures 6(b) and 6(d), the boundary-layer flow over the stern, upstream of the propeller, is similar in terms of thickness and unsteadiness between the two propeller diameters. The only observed difference is the low-speed region between the lower rudders which appears to be somewhat faster with the 4.0 m propeller. At the propeller plane, the 5.0 m propeller easily ingests the hull boundary layer giving the outer portion of the blades near the tips exposure to higher flow speeds. In contrast, the 4.0 m propeller almost manages to ingest the full hull boundary layer but some pockets of lower-speed flow do propagate outside of the propeller slipstream. The higher loading of the 4.0 m propeller coupled with the root profile modification generates faster mid-span flow speeds and causes the hub boundary-layer flow immediately behind the blades to significantly reduce in thickness and increase in speed relative to the 5.0 m propeller. Consequently, the hub vortex generated is very compact in size but has a core velocity of similar magnitude to the 5.0 m propeller. The character of the hub vortex and the propeller slipstream in general, changes dramatically further downstream. For the 4.0 m propeller configuration, increasing flow instabilities cause the hub vortex and propeller wake to rapidly break down. However, the 5.0 m propeller maintains a coherent hub vortex and propeller slipstream, which dissipates at a gradual rate.

Figure 7 shows instantaneous cross sections of the axial velocity at \( x/L \approx 0.93 \), 0.98, 1.03 and 1.05 for both propellers, with the upper and lower subset of figures belonging to the 4.0 m and 5.0 m propeller configurations, respectively. In the first column of figures, representing a plane upstream of the propeller at \( x/L \approx 0.93 \), the four rudder wakes are visible which divide the flow into four quadrants. The boundary layers in the lower and side quadrants are similar, whereas the flow in the upper quadrant exhibits a different pattern and higher flow velocities due to interactions with the fin horse-shoe vortex, fin wake and accelerated flow over the rear part of the casing. Within the propeller disc plane, at \( x/L \approx 0.98 \), flow is markedly different between the two propellers. As previously indicated, the boundary-layer flow near the hub is much thicker for the 5.0 m propeller and the blade tips see a more uniform, higher-speed flow throughout an entire rotation leading to tip vortices on each blade being more uniform in size. In contrast, the blade tips of the 4.0 m propeller which pass through the upper quadrant produce larger tip vortices relative to the remaining blades, which have smaller tip vortices. Due to the hull wake, the blade tips located outside of the upper quadrant see axial flow velocities which are lower than those experienced by the 5.0 m propeller. Furthermore, the presence of the horse-shoe vortices and the wake deficit from the rudders is more prominent in the 4.0 m propeller configuration. The downstream cross sections at...
Figure 5. Vortical and wake structures at 4.6 kt, visualised with iso-surfaces of $\lambda_2 = -0.05$: (a) wide view of the entire boat with the 5.0 m diameter propeller; (b) flow structures around the rudders and in the propeller near-wake for the 4.0 m propeller; and (c) flow structures around the rudders and in the propeller near-wake for the original 5.0 m DSTG 057-1 propeller.

$x/L \approx 1.03$ and $x/L \approx 1.05$ have similar characteristics for each propeller but are different between propellers. The 4.0 m diameter propeller exhibits stronger remnants of the horse-shoe vortices and fin wake in comparison with the 5.0 m propeller. In addition, the hub vortex is more compact with the 4.0 m propeller and has a uniformly high (clipped) velocity across the propeller disc due to its higher loading relative to the 5.0 m propeller.

Figure 8 compares the predictions of the time-averaged normalised axial velocity, $\langle u \rangle / U_\infty$, the normalised axial rms velocity fluctuations, $u_{rms} / U_\infty$, and the pressure coefficient, $C_p = \langle p \rangle / \left( \frac{1}{2} U_\infty^2 \right)$, at seven axial locations from $x/L = 0.850$ to $x/L = 1.110$ for both propeller configurations as labelled in the figures. The time-averaging is performed over nine or more propeller revolutions as listed in Table 2. Figure 8(a) shows that upstream of the propeller $\langle u \rangle / U_\infty$ is essentially unaffected by a change in propeller diameter. However, major differences are apparent in the propeller slipstream, with the 4.0 m propeller exhibiting higher velocities across the propeller disk area yet maintaining a hub vortex deficit of similar magnitude relative to the 5.0 m propeller. In Figure 8(b), upstream of the propeller $u_{rms} / U_\infty$ is again largely unaffected by the propeller diameter. Downstream of the propeller, the 4.0 m configuration yields values that are similar but lower in magnitude across the blade relative to the 5.0 m configuration with two exceptions: near the propeller tip vortices, the 4.0 m propeller produces slightly higher values; and in the vicinity of the hub vortex, the 4.0 m propeller produces significantly higher values immediately behind the hub than the 5.0 m propeller. Figures 8(a) and 8(b) both show that the tip velocity of the 4.0 m propeller is higher than that of the 5.0 m propeller. Figure 8(c) shows the 4.0 m propeller generates a significantly lower $C_p$ value relative to the 5.0 m propeller.

The mean pressure coefficient on the surface of both propellers is presented in Figure 9. The 5.0 m propeller has very low pressures on the suction side, which are located from mid-span to the tip of the blade, and between about 25% to 75% of the chord length. Then moving from mid-span towards the blade root, pressure increases. The trailing edge at mid-span also has low pressures, whereas the root and tip regions have higher pressures. High pressures are found at the leading edge along all of the blade except for the tip region, indicating the presence of a negative incidence angle. The $C_p$ on the pressure side becomes increasingly lower when moving from mid-span to the root, and more so from mid-span to the tip, although this is present mainly across the first half of the chord from the leading edge. From approximately mid-chord to the trailing edge of the blade sections, $C_p$ increases to near unity. The overall result is a highly-loaded mid-span region along with lightly loaded tip and hub region; to be expected for a wake-adapted design, even for one not tailored for this boat. The 4.0 m propeller, being a scaled version of the same design, has similar characteristics to the 5.0 m design, but with the highly loaded areas expanding in size - especially near the leading edge and tip on the suction side. The loss of part of the original root section due to scaling results in the new root of the blade (equivalent to the low-mid span section of the 5.0 m propeller) being more highly loaded. This, coupled with the higher overall loading, leads to the stronger hub and tip vortices identified earlier with the 4.0 m propeller.
Figure 6. Contours of instantaneous axial-velocity, $U_x$, on the centreline plane ($y = 0$) for: (a) & (b) the 4.0 m-scaled DSTG 057-1 propeller; (c) & (d) the original 5.0 m propeller. (a) & (c) show wide views of the instantaneous flow around the bow and mid-section of the boat; (c) & (d) show zoomed views of the flow around the stern region including the propeller and near wake.

Figure 7. Contours of instantaneous axial-velocity, $U_x$, on four $y-z$ planes at $x/L \approx 0.93$, $x/L \approx 0.98$, $x/L \approx 1.03$ and $x/L \approx 1.05$. The top row of plots represents data from the simulation with the 4.0 m-scaled DSTG 057-1 propeller at 4.6 kt; the bottom row of plots show data from the simulation with the original 5.0 m diameter propeller at 4.6 kt. The colour scale used is identical to that in Figure 6.
Figure 8. (a) Profiles of the normalised mean axial velocity, $\bar{u}/U_\infty$; (b) the normalised root-mean-square of axial velocity fluctuations, $u_{r.m.s.}/U_\infty$; and (c) the pressure coefficient, $C_p$. Profiles are shown at seven locations in the axial direction (as labelled on the plot). The spacing between the profiles is not to scale. $-$, $D=4.0$ m; $-$, $D=5.0$ m.
CONCLUSIONS

A near-wall-modelled LES approach has been used to simulate the flow around a full-scale, fully-appended Jou-bert generic submarine equipped with either a 4.0 m or 5.0 m diameter 7-bladed propeller under straight-ahead, self-propulsion conditions at 4.6 knots. In terms of general characteristics, the flow field shows that strong horseshoe vortex systems are predicted to form around the base of the fin and each of the rudders, these interacting with each other and the highly energetic fin wake over the stern. Highly resolved vortices are also seen emanating from the rear of the fin- and rudder caps. For both cases, the propeller wake flow appears well captured, with tight tip vortices propagating downstream before starting to lose coherence.

Since the 4.0 m propeller is a scaled version of the 5.0 m propeller with some of the blade root being removed, the rpm was increased by around 36% to produce the same level of thrust as the 5.0 m propeller. This leads to higher blade loadings and stronger hub and tip vortices on the 4.0 m propeller. However, the propeller slipstream flow illustrates higher levels of unsteadiness associated with the 4.0 m propeller wake, leading to a rapid breakdown of its hub and tip vortices, whereas the 5.0 m propeller maintains a coherent and slowly dissipating wake.

REFERENCES


**DISCUSSION**

**Question from Rickard E. Bensow**
How did the force balance in self-propulsion change when going from RANS to LES?

**Author’s closure**
The focus of the paper was to provide insights into the flow physics using LES with self-propulsion conditions, and hence forces, only computed with RANS. However, forces were monitored on the LES solutions and, unlike the RANS solutions, they do indeed exhibit a mismatch in axial forces. More specifically, the propeller thrust is of greater magnitude than the hull resistance. With regards to propeller thrust, which is dominated by pressure rather than viscous forces, the difference in magnitude between the RANS and LES solutions is within a few percent. However, as reported in earlier work (Norrison et al., 2016), the hull resistance values from the LES solutions are about one third of that predicted by RANS. This discrepancy is attributed to viscous effects for wall bounded flows which are challenging to accurately predict using LES. We are continuing to work on reducing the viscous force under-predictions associated with the hull resistance for the full-scale simulations.

**Question from Amadeo Moran Guerrero**
Did you try DES approach?

**Author’s closure**
Detached Eddy Simulation (DES) is a hybrid approach combining RANS and LES as a means to reduce the high computational cost of a pure LES which was used in this study. We have not employed DES for the cases studied.

**Question from Nobuhiro Hasuike**
In the case of propeller design at same shaft speed, how do you think about how to change the tip vortex and hub vortex with different propeller diameters?

**Author’s closure**
The purpose of this paper was to explore the complex interaction of unsteady (temporally varying) wake structures and to observe how they dissipate downstream of the boat. The scope of the work fixed the boat speed, while allowing the shaft speed to vary to achieve self-propulsion conditions. The alternative, as suggested, is to fix the shaft speed and allow the boat speed to vary for a given propeller diameter. This would result in a different Reynolds number, and hence flow conditions, around the boat which would change the behaviour of the flow, including varying the hub and tip vortices. It is uncertain how significant the variation would be but this would certainly make for a very interesting study in the future.