Computational Fluid Dynamics Simulation of a Rim Driven Thruster

Aleksander J Dubas, N. W. Bressloff, H. Fangohr, S. M. Sharkh (*University of Southampton*)

Abstract

An electric rim driven thruster is a relatively new marine propulsion device that uses a motor in its casing to drive a propeller by its rim and the fluid dynamics associated with their operation have not been fully investigated. There are many interacting flow features that make up the flow field of a rim driven thruster that pose a number of challenges when it comes to simulating the device using computational fluid dynamics. The purpose of this work is to develop a computational fluid dynamics solution process that accurately simulates features including vortex generation and behaviour, radial pumping and rotor-stator interaction while attempting to minimise computational costs. This will enable the method to be used to calculate an objective function, typically the thrust or propulsive efficiency of the device, in a design optimisation study. Implementation within a design optimisation study also requires the numerical methods to be easily repeatable and robust in both mesh generation and solution.

Mesh generation was performed using snappyHexMesh, a meshing program that is part of OpenFOAM, and a thorough mesh verification procedure has been conducted. Validation of the computational fluid dynamics solution of a standard series propeller, as a baseline case with good experimental data from MARIN, using the open source Reynolds-Averaged Navier-Stokes solver MRFSimpleFoam (part of the OpenFOAM software) has been performed. Results show a great sensitivity to computational domain size that suggest that similar previous works may have used an insufficient domain size. In particular, it is shown that a number of boundary conditions may be used if the domain is large enough. Also, comparisons are made between the Re-Normalisation Group (RNG) k-ϵ and k-ω Shear Stress Transport (SST) turbulence models (the most widely reported models in the literature), and the k-ω SST model is found to be robust due to its better handling of the separation that occurs at low propeller advance ratios. Validation against experimental data for the standard series propeller shows good agreement to within 5%.

The validated solution method is then applied to a rim driven thruster and key design areas are highlighted by the results. The rim is found to be an important region of the flow, the drag on which comprises almost half of the torque losses in the device. Interaction between the rotors and the stators is also a key area, with both thrust and torque changing as the position of the blades is varied.

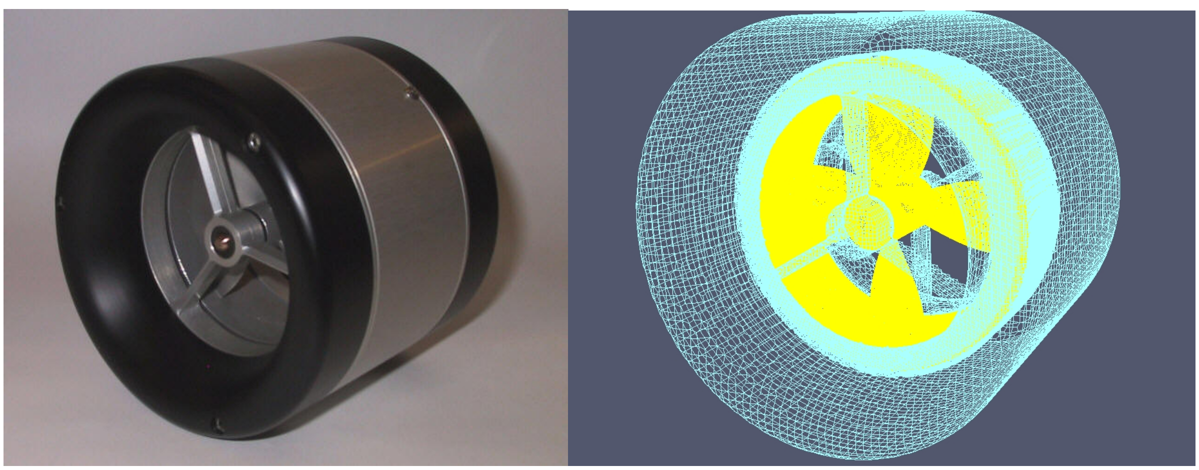
1 Introduction

An electromagnetic marine propulsion device that uses a brushless DC motor built in to its casing and propeller rim has recently been developed [1]. This rim driven thruster, pictured in Figure 1a, has many interacting flow features that make up its flow field, posing a number of challenges when attempting to simulate the device computationally. While there have been previous attempts at simulating the flow using panel methods [2], these lack the fidelity and modelling capabilities of the RANS (Reynolds-Averaged Navier-Stokes) methods here.

The aim of this work is to gain insight into the the flow through a rim driven thruster and the interaction of its flow features, and to use this insight to help optimise the design to improve its propulsive efficiency. Multiple design iterations require a robust solution method that accurately simulates the flow features with minimal computational costs.

2 Methods

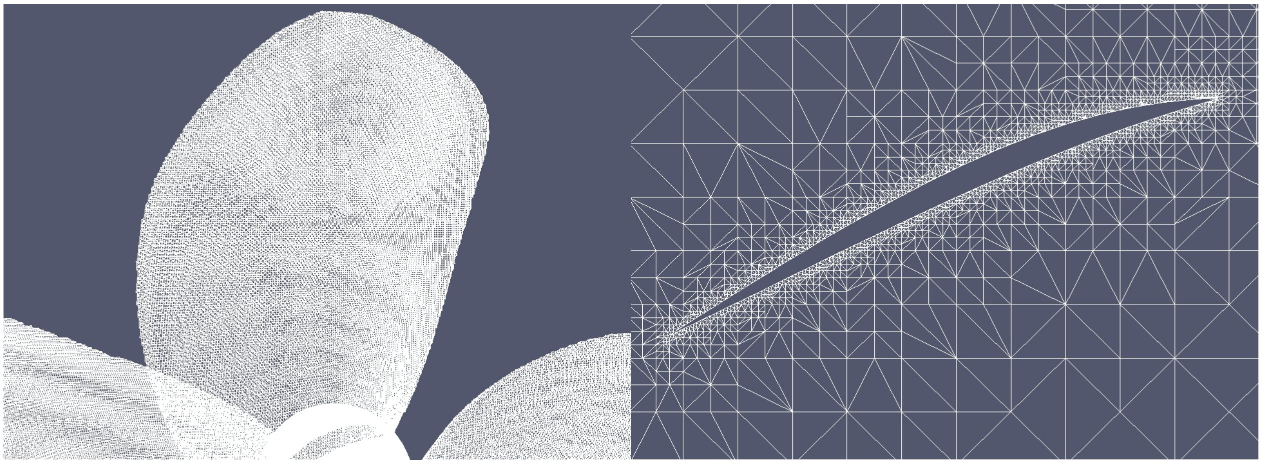
The meshes in this work were generated using the OpenFOAM tool snappyHexMesh, after the baseline mesh and computational domain were initially created using blockMesh, another OpenFOAM program. Steady state simulations were then performed using the OpenFOAM solver MRFSimpleFoam and post processed using Paraview.



(a) Photograph. (b) Computational mesh.

Figure 1: Pictures of the rim driven thruster.

For the purposes of verification and validation, initial simulations modelled a Wageningen B4-70 propeller for which there is published experimental data [3]. The baseline test case used a 70mm propeller advancing at one metre per second and revolving at 3000rpm, similar to the operating conditions of the rim driven thruster. The verification process involved a mesh dependency study, where the surface resolution was increased, while the layer mesh was kept constant to fix y^+ (the non-dimensional distance from the wall of the first layer of cells), until the results did not change any further. With a baseline cell size of 17.5mm,



(a) Surface mesh. (b) Blade passage mesh at r/R = 0.7.

Figure 2: Mesh visualisations for the Wageningen B4-70 propeller case.

one quarter of the diameter, the minimum surface resolution on the blade was found to correspond to a surface refinement level of six. This gave a cell count of roughly 360,000 cells with an average y^+ of approximately 20 across the surface of the blade. The resulting mesh for the Wageningen B4-70 propeller can be seen in Figure 2 and the meshing of the rim driven thruster is depicted in Figure 1b, where the yellow parts are rotational and the blue parts are static.

The second stage of the verification process aimed to find a computational domain size at which any further increase in domain size did not affect the results. As shown in Figure 3, the minimum required computational domain width was found to be six propeller diameters, considerably larger than the domain sizes used in similar works on propeller modelling[4, 5, 6].

To ensure that the solution procedure had converged to a result, a number of metrics were monitored over simulation time (one second per timestep). The magnitude of both the residuals (Figure 4a) and timestep continuity errors (Figure 4b) were checked and were found to have fallen below 10^{-4}. A further check was made by plotting the variables of interest, force (Figure 4c) and torque (Figure 4d), to see if there was any oscillation or divergence. As shown in Figure 4, the force and torque reach a steady value in a relatively few number of iterations compared to the number of iterations before the residuals and timestep continuity errors appear to converge. Thus, for the purposes of a design optimisation study, it would be possible to use less iterations to reduce the computational cost of each calculation of an objective function based on the z force or torque.

Two turbulence models were tested, first the RNG k-ϵ turbulence model as the most widely used in the literature[4, 7, 8]. Later the k-ω SST turbulence model was tested for its more robust handling of low speed separation after some difficulty was experienced in solution convergence at low advance ratios. The k-ω SST model was found to be the more robust of the two models although both showed good agreement with experimental data.

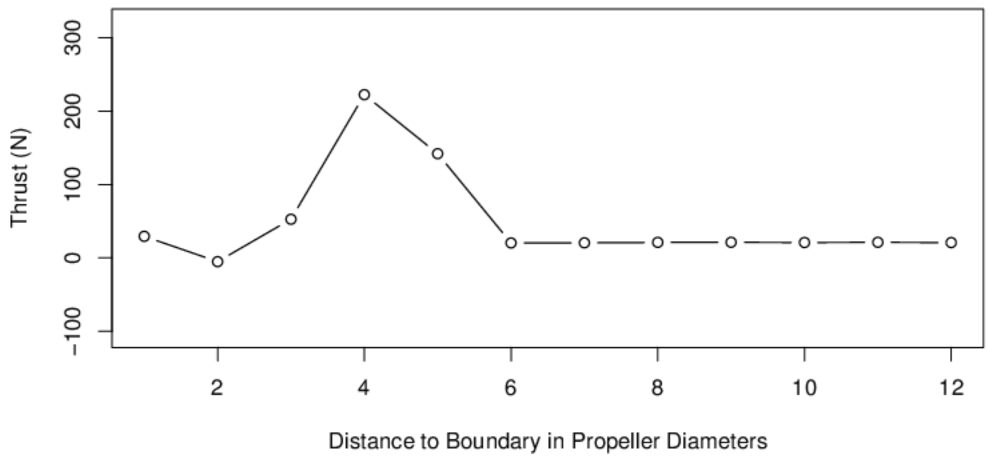


Figure 3: Computational domain size study showing the effect of distance to domain walls on thrust calculated.

3 Results

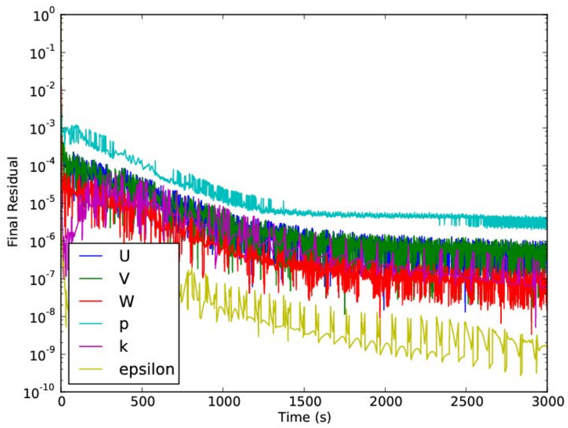
Figures 5 and 6 show the validation results for the RNG k-ϵ and k-ω SST turbulence models respectively. The good agreement with experimental data gives confidence in the simulation results, although there are some differences. This could be attributed to subtle variations in the computational and experimental geometries at the blade tip, which occur due to the difficulty of generating the rounded edge of the blade tip of an open water propeller. There could also be experimental errors that affect the agreement with the validation data.

After validation of the method, a test model of a rim driven thruster with an experimental blade configuration, provided by TSL Technology Ltd., was simulated and results were compared against experimental towing tank data. While the agreement with the results was not as good as for the open water propeller, some insight was gained into the flow features, and the difference between experiment and simulation is thought to be predominantly due to the 'frozen-rotor' handling of the unsteady rotor-stator interaction by MRFSimpleFoam.

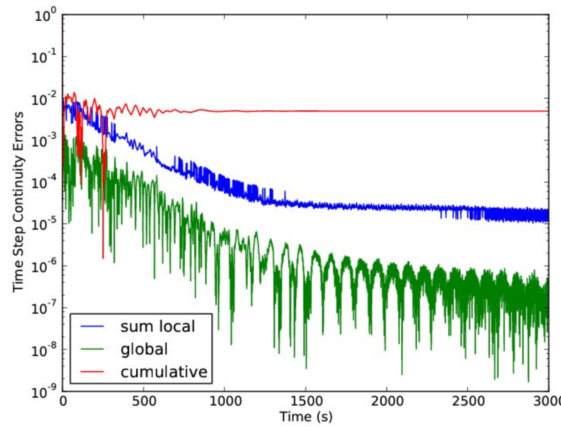
From the results for the rim driven thruster, it can be seen from the pressure distributions in Figure 7 that there is high negative pressure at the tips of the blades, close to the rim. Not only could this low pressure region lead to problems with cavitation at high blade loading, but also suggest that the pitch distribution of the blades could be better optimised to account for the effect of the duct and stators on the inflow.

Thruster Part	Thrust	Torque
Casing	-1.18	-0.104
Stators	-8.31	-0.256
Blades	19.40	0.275
Rim	2.53	0.161

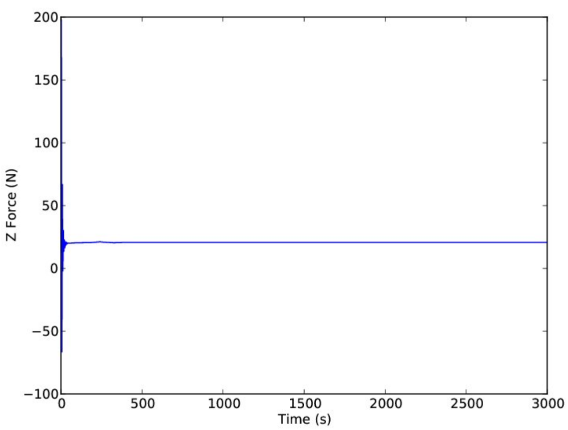
Table 1: Breakdown of the contributions of each part to the total forces.



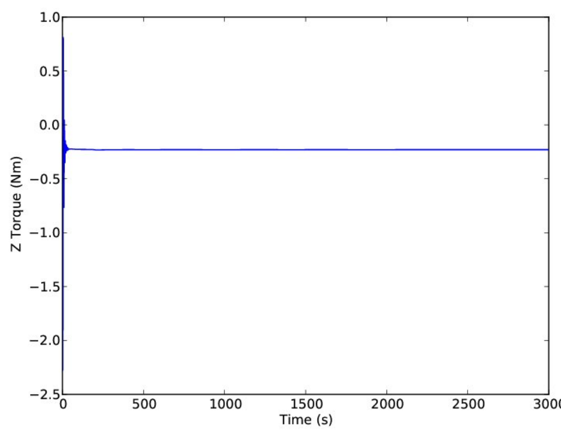
(a) Final residuals.



(b) Time step continuity errors.



(c) Force results.



(d) Torque results.

Figure 4: Solution convergence.

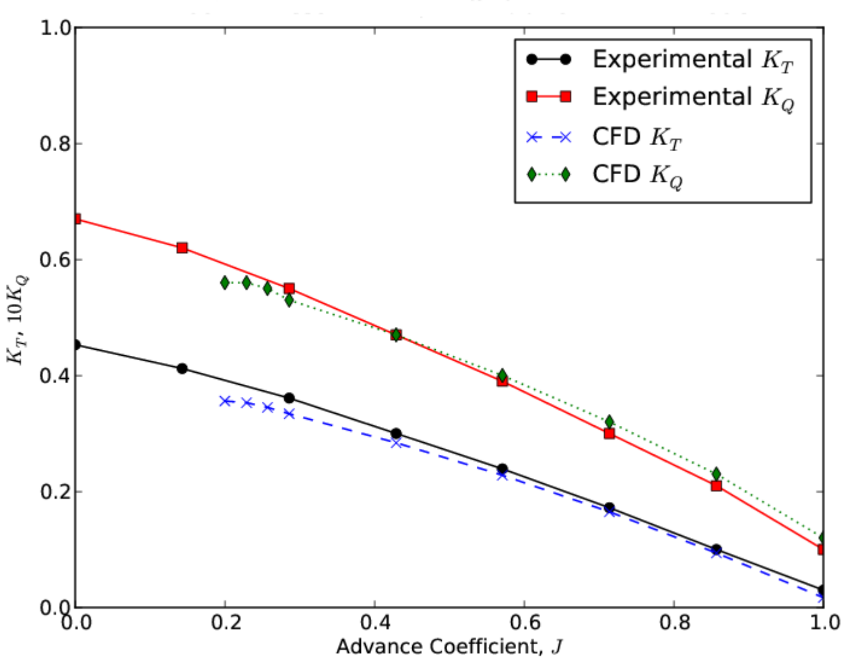


Figure 5: Validation against experimental data for the Wageningen B4-70 propeller using RNG k-ϵ turbulence model.

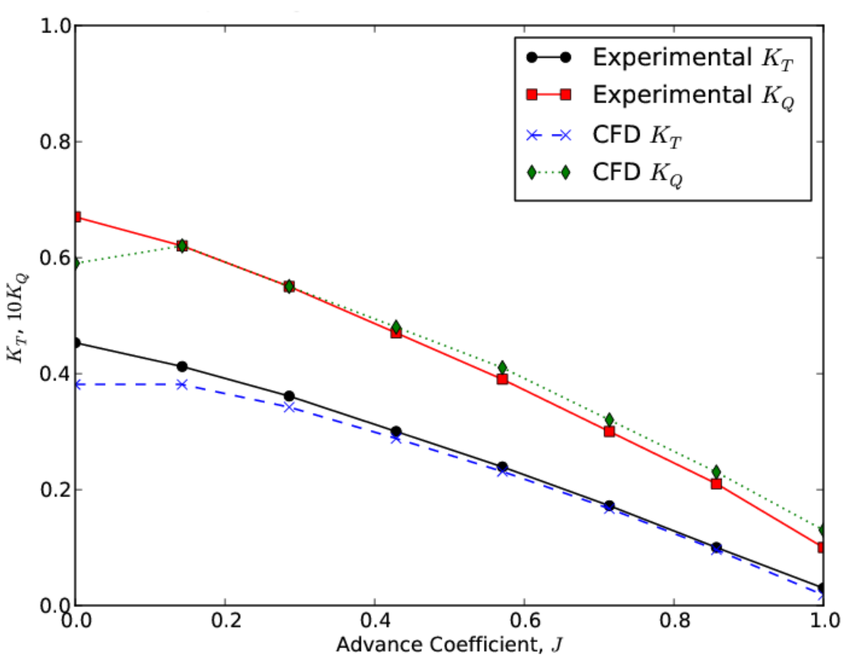


Figure 6: Validation against experimental data for the Wageningen B4-70 propeller using k-ω SST turbulence model.

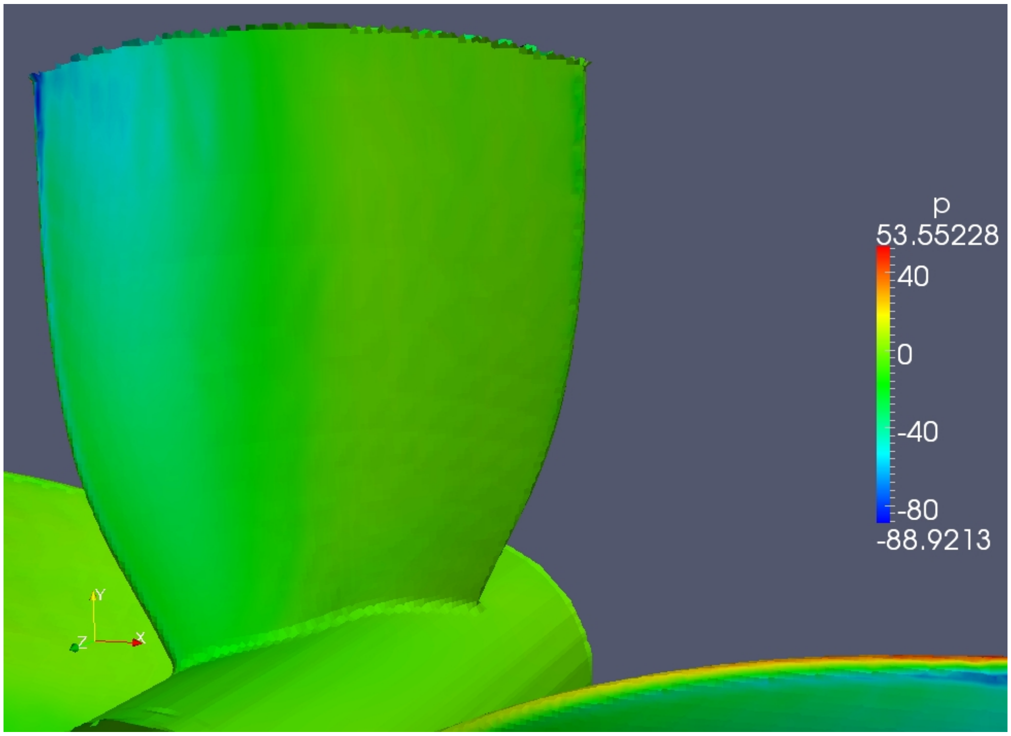


Figure 7: Visualisation of the pressure distribution on the rim driven thruster blades at an advance ratio of 0.2857, at 3000rpm.

Another interesting result from the rim driven thruster is the breakdown of force contributions from each individual part. Table 1 shows how each part of the thruster affects the performance, the interesting component of which is the rim. It is shown that the rim contributes both to the thrust, from the pressure difference across it, and also to the torque, comprising about one third of the hydrodynamic losses in the device. A further assessment of the source of the torque on the rim shows that the force is primarily from viscous shear.

Blade Angle	K_T	K_Q	η
0	0.198	0.0185	0.487
15	0.216	0.0176	0.561
30	0.204	0.0116	0.798
45	0.202	0.0176	0.521
60	0.209	0.0253	0.376
75	0.170	0.0227	0.342

Table 2: Thrust, torque and efficiency coefficients at a range of blade angles, relative to the stators.

A preliminary study was made into the potential effect of the rotor-stator interaction by looking at steady state simulations at different blade angles (periodic every 90 degrees as it is a four bladed propeller). The results from this study shown in Table 2, while not being a exact simulation of the unsteady problem, gives a clear indication of the importance of the rotor-stator interaction, as the efficiency is shown to vary by over 100% between blade angles at 30 and 75 degrees.

4 Discussion

One of the major problems encountered in the validation procedure described above was with convergence at low advance ratios when using the RNG k-ϵ turbulence model. The critical advance ratio below which solutions stopped converging reduced when rotational speed, and consequently Reynolds number, was increased. At lower advance speeds, the effective angle of incidence is larger, thus it is thought that the cause of the convergence issues is the onset of separation. The k-ω SST model was tested to see if it handled the separation better and the converged results at low advance ratios achieved confirmed this hypothesis.

However, Figure 6 still shows a large discrepancy between simulation and experimental results at low advance speeds. Previous work over a similar range of advance ratios shows a similar result[8] and thus further experimental investigation is needed to ascertain the source of this difference.

One of the aims is to improve the design performance of the rim driven thruster device, with the results so far highlighting the key areas for investigation. First, the blade pressure distribution obtained in Figure 7 suggests that there is scope for optimisation of the blade pitch and section as the loading on the propeller appears to be biased towards the leading edge of the blade tip. The significant portion of torque attributed to viscous shear on the rim indicates that the rim and the gap between the rim and casing is a key area for minimising the losses and thus improving the efficiency. There is also the rotor-stator interaction to consider as the results in Table 2 can be interpereted to imply a source of huge variability caused by this phenomenon. While the variation of thrust in Table 2 has a range of 27% of the minimum value, the variation of torque has a range of 96% of the minimum value leading to a maximum difference in reported propulsive efficiency of 133%. This is further backed up as an important area for investigation by the significant amount of drag (or loss of thrust) on the stators reported in Table 1. Unfortunately as all the above areas are interacting areas of the flow, it would be best to optimise all the variables simultaneously, though computational expense may limit the extent to which this can be done.

5 Conclusion

The computational fluid dynamics of rotational devices was performed using MRFSimpleFoam, and good agreement with experimental data was found for open water propellers. Application of this solver to a rim driven thruster has given some insight into the flow through the device, highlighting key design areas of the rim gap, blade pitch distribution and stators. The steady solution method using MRFSimpleFoam is limited in its application to rotor-stator interaction and further work is needed for a more accurate model and problem set-up. There are a number of methods that could be used to investigate the rotor-stator interaction. Either the stators could be removed from the computational model and considered separately, or they could be smoothed through the use of a mixing plane, or simulated accurately using unsteady computational fluid dynamics.

At the bollard pull condition, there is a significant difference between the experimental and computational results. Further investigation into the source of this difference is required. Additional experimental data would help and gathering of field data, such as velocity or pressure distributions, would improve the confidence in the validation of the computational fluid dynamics method.

6 Acknowledgements

This work was supported by an EPSRC Doctoral Training Centre grant (EP/G03690X/1). The thruster geometry and some help with modelling the open water propeller were provided by TSL Technology Ltd.

References

[1] S. M. Sharkh, S. R. Turnock, and G. Draper, "Performance of a tip-driven electric thruster for unmanned water vehicles," in *Proceedings of the International Offshore and Polar Engineering Conference*, vol. 2, pp. 321–324, 2001.

[2] A. W. Hughes, S. R. Turnock, and S. M. Sharkh, "CFD modelling of a novel electromagnetic tip-driven thruster," in *Proceedings of the Tenth International Offshore and Polar Engineering Conference*, 2000.

[3] G. Kuiper, *The Wageningen Propeller Series*. MARIN, 1992.

[4] S. Huang, X. yuan Zhu, C. yu Guo, and X. Chang, "CFD simulation of propeller and rudder performance when using additonal thrust fins," *Journal of Marine Science and Application*, vol. 6, no. 4, pp. 27–31, 2007.

[5] S. H. Rhee and S. Joshi, "Computational validation for flow around a marine propeller using unstructured mesh based Navier-Stokes solver," *JSME International Journal, Series B*, vol. 48, no. 3, pp. 562–570, 2005.

[6] R. E. Bensow and G. Bark, "Implicit LES predictions of the cavitating flow on a propeller," *Journal of Fluids Engineering*, vol. 132, 2010.

[7] W. Lam, D. J. Robinson, G. A. Hamill, S. Raghunathan, and C. Kee, "Simulation of a ship's propeller wash," in *Proceedings of the Sixteenth (2006) International Offshore and Polar Engineering Conference*, pp. 457–462, 2006.

[8] T. Abramowski, K. Zelazny, and T. Szelangiewicz, "Numerical analysis of influence of ship hull form modification on ship resistance and propulsion characteristics - part III - influence of hull form modification on screw propeller efficiency," *Polish Maritime Research*, vol. 17, no. 1, pp. 10–13, 2010.