A. Yücel Odabaşı Colloquium Series
3rd International Meeting on Progress in Propeller Cavitation
And its Consequences: Experimental and Computational Methods for Predictions
14th – 16th November 2018, Istanbul, Turkey

Tip Vortex Cavitation Simulation of a Propeller in a Gate Rudder® System

Naz Yilmaz[1*], Serkan Turkmen[2], Batuhan Aktas[1],
Patrick Fitzsimmons[1], Noriyuki Sasaki[1], Mehmet Atlar[1]
[1]*Department of Naval Architecture, Ocean and Marine Engineering, University of Strathclyde, Glasgow G4 0LZ, UK*
[2]*School of Marine Science and Technology, University of Newcastle, Newcastle upon Tyne NE1 7RU, UK*

Abstract: The GATE RUDDER® system is a novel propulsion arrangement or Energy Saving Device (ESD) inspired by the new concept of elementary propulsive efficiency and its optimization in a ship's wake to recover more energy. The performance of a GATE RUDDER® system in the hull wake, therefore, is important not only for the efficiency but also from the cavitation, noise and vibration point of view. The World's first gate rudder was installed on a 2,400 GT container ship in 2017 in Japan. By using the data associated with this vessel and other model test data with different ships, this paper explores the differences on the efficiency and cavitation performance of a conventional rudder and propeller system with the GATE RUDDER® system using Experimental and Computational Fluid Dynamics (EFD and CFD) approaches. There is specific emphasis on the accurate simulation of the tip vortex cavitation of the propeller in both rudder systems which has been modelled by using Yilmaz's recently developed advanced adaptive mesh refinement approach. The results of the CFD simulations are compared with the results of the model tests conducted in the Emerson Cavitation Tunnel and the full-scale experiences with the above-mentioned container vessel as discussed in the paper.

Keywords: Gate Rudder, Propeller Cavitation, Cavitation Tunnel Tests, Tip Vortex Cavitation, CFD, Adaptive Mesh Refinement

1 INTRODUCTION

In order to improve the energy efficiency of ships, and hence to achieve targeted carbon emission (e.g. EEDI regulations by IMO), various technological and operational solutions have been studied by the maritime industry. These solutions recently included the developments of various novel Energy Saving Devices (ESD) applied on the underwater hull and renewable energy saving devices onboard, using alternative fuel sources and sophisticatedly optimized hull forms. Such ESD solutions still, developed at model scale have their challenges to prove their effectiveness on full-scale ships. Although many ESDs already exist and some new types are still being introduced, their effectiveness, especially in full-scale, need to be investigated and proven further by using preferably Computational Fluid Dynamics (CFD) and Experimental Fluid Dynamics (EFD) methods under operational conditions. This is not only at the development stage of these devices but also at the selection stage for a particular ship type.

The selection of an effective ESD technology for a ship amongst the wide range of solutions may be made based on the personal preference of the so-called experts of the company, often due to a bias against a particular type of technology, rather than using a sophisticated tool. However, the selection of such devices must be carried out using scientifically proven methods, preferably by using the CFD and supporting EFD approaches as well as by conducting a techno-economic feasibility assessment of the selected technology on a particular marine system, taking into consideration the payback time, maintenance requirements and expenses, retrofitting, etc., for a given operational profile.

Within the framework of a newly introduced ESD system, the main purpose of this study is to utilize the EFD and CFD methods to demonstrate the effectiveness of a novel ESD, which is called the "GATE RUDDER®" system, with a specific emphasis on the cavitation and noise performance of this ESD in comparison to a conventional rudder-propeller system.

The GATE RUDDER® system is a new and innovative ESD technology for ships to propel and steer them more efficiently. As opposed to a conventional rudder, which is behind a propeller, the GATE RUDDER® has two rudder blades with asymmetric sections which are located alongside the propeller, and each blade can be controlled independently. The two rudder blades, encircling the propeller at the top and sides, provide a duct effect and hence produce additional thrust as opposed to the additional drag of a conventional rudder behind the

* *Corresponding author e-mail: naz.gorener@strath.ac.uk*

propeller. See Figure 1 for comparison of the conventional rudder and the GATE RUDDER® system on two sister vessels. The independent control of the two rudder blades also provide effective control of the propeller slipstream and hence steering, Sasaki et al (2015). Thus the GATE RUDDER® system presents not only more propulsive efficiency but also higher maneuverability. In addition to these two major advantages of the GATE RUDDER® system, there are other claimed performance superiorities of this system, including reduced vibrations, as reported through full-scale performance trials (Sasaki et al., 2018), and which may be associated with the reduced cavity volume on the blades as well as the tip vortex in the propeller slipstream. These effects require investigations.

In order to cast light on the cavitation performance of the GATE RUDDER® system, this paper investigated the cavitation performance of the GATE RUDDER® system in comparison with that of a conventional rudder-propeller system, for the first time, by using the CFD and EFD approaches. The rudder systems used in this investigation for the conventional and GATE RUDDER® arrangements are based on the two sister vessels recently built in Japan and have been in service since 2016 (conventional rudder-propeller system) and 2017 (GATE RUDDER® system). The cavitation tunnel tests for the EFD investigations were conducted in the Emerson Cavitation Tunnel at the University of Newcastle upon Tyne.

Following this introductory section, the paper begins with the background to the GATE RUDDER® concept and details of the rudder and propeller arrangements of the two sister ships as presented in §2. The details of the rudder and propeller models, cavitation tunnel test set-up, and test conditions and procedures, which formed the basis for the EFD investigations, are presented in §3. The details of the CFD simulations of the propeller with the conventional and GATE RUDDER® in the cavitating conditions are presented in §4 to form a basis for the EFD investigations. This is followed by the presentation and discussion of the comparative results for the cavitation simulations based on the CFD and EFD approaches for the Conventional and GATE RUDDER® system in §5. Finally, the concluding remarks with future work are presented in §6.

2 GATE RUDDER ® CONCEPT

The rudder is one of the resistance components of the ship. The main purpose of the GATE RUDDER® propulsion system is to replace the resistance source (of a conventional rudder system) with a thrust source (like a duct) to achieve higher propulsive efficiency. With this idea, the rudder may become an ESD placed alongside the propeller instead of behind the propeller to simulate the duct effect of a ducted propeller but with additional maneuverability capability by independently moving the two rudder blades to control the propeller slipstream in contrast to the nozzle of a fixed ducted propeller. The GATE RUDDER® arrangement also reduces the viscous energy loses created by the hull boundary layer and the wake flow more effectively than the traditional rudder-propeller arrangement, Sasaki et al (2018).

In a similar way, although many ideas and applications exist to combine a rudder and a propeller, such as podded propulsion systems, steerable ducted propellers and so on (e.g. Carlton, 2012), these propulsion systems generally work with limited applications in the full scale without high propulsive performances and maneuverability abilities. Whereas the GATE RUDDER® propulsion system has a flexibility that can be applied to a new design as well as a retrofit system to almost many types of conventional vessel where the conventional rudder-propeller system is used.

As reported in Sasaki et al (2018) the GATE RUDDER® propulsion system originated in Japan and has been further developed in the UK through CFD and EFD studies since 2014. Based on these developments, the first GATE RUDDER® propulsion system was applied on a 2400 GT container ship and the full-scale sea trials were carried out on November 2017 in Japan. The performance gain expected from the application of this novel ESD was demonstrated by the comparison of these trial results with of the results of her sister container vessel of the similar size and characteristics but fitted with a conventional flap rudder-propeller system that was delivered one year before. Both vessels currently operate in the same route in Japan between Hokkaido and Yokohama. Figure 1 shows the propeller and rudder arrangement of these two sister vessels, which are indicated as Ship A (with the conventional rudder-propeller system) and Ship B (with the GATE RUDDER® propulsion system), respectively while Table 1 presents their main particulars.

The analyses of the sea trials data conducted in the same geographic region of Japan with the two vessels within a year interval and those of the voyage data on the same service routes indicated that the container vessel with the GATE RUDDER® system can save abt. %14 more fuel over the vessel with the conventional rudder-propeller system. It was noticed that 8-10% of this attractive energy saving was confirmed by the CFD and EFD studies while the remaining saving can be attributed to the scale-effect associated with the powering estimation with the GATE RUDDER® system as demonstrated in recent studies, (Sasaki et al., 2018).

Based on the experiences during the sea trials and following onboard experiences of the both vessels' captains during service, it was noticed that the vessel with the GATE RUDDER® experienced less propeller excited vibrations with quieter aft end characteristics compared to those of the vessel with the conventional rudder-propulsion system. Based on these findings, as the main objective of this paper, it was decided to explore the cavitation and noise characteristics of the two propulsion systems using the detailed CFD and EFD investigations based on the aft end arrangement and operating conditions of these two sister vessels and by using a state-of-the-art commercial CFD tool and model tests conducted in a medium-size cavitation tunnel with simulated wakes.

Table 1. Main Particulars of two sister ships

Vessel particulars	Ship A (Conventional Rudder)	Ship B (GATE RUDDER®)
Loa (m)	111.4	111.4
Lpp (m)	101.9	101.9
B (m)	17.8	17.8
D (m)	8.5	8.5
d (m)	5.24	5.25
Main Engine	3309kW/220rpm	3309kW/220rpm
Prop. Dia (m)	3.48 (CPP)	3.30 (CPP)
Draft of Sea Trial (m)	4.30	4.30

Figure 1. Conventional Flap-Rudder (Top) vs. GATE RUDDER® (Bottom)

3 CAVITATION TUNNEL TESTS

3.1 Cavitation Tunnel and test set-up

Although comprehensive experimental tests were conducted with a 2m model during the GATE RUDDER® developments in towing tanks and circulation channels, which involved powering, maneuvering and seakeeping, no cavitation tunnel tests were conducted until this study explored the comparative cavitation and underwater radiated noise (URN) characteristics of the GATE RUDDER® propulsion system.

The aft end and propeller arrangements of the conventional rudder (without the flap) and GATE RUDDER® systems were represented with the model rudders and propellers of the two vessels with a scale ratio of 13.2 and fitted downstream of the H33 K&R dynamometer of the Emerson Cavitation Tunnel (ECT) of Newcastle University. ECT is a medium-size facility with a measuring section of 3.1m x 1.21m x 0.8m (L x B x H) with other details as shown in Figure 2 and reported in (Atlar, 2011).

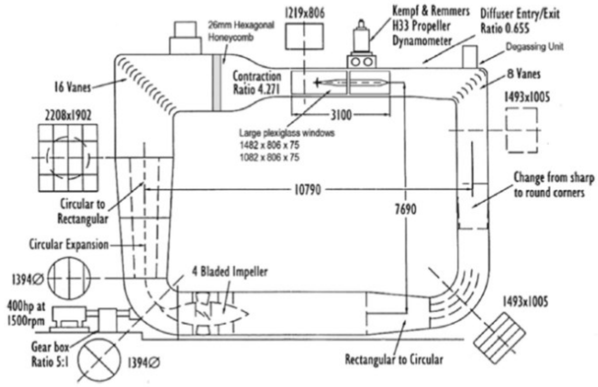

Figure 2. Emerson Cavitation Tunnel

While the measuring section of the ECT usually allows a reasonable size dummy hull with a properly scaled aft end arrangement, in this investigation, a simple wake simulation arrangement was used due to time restrictions. In this arrangement, the wake of the H33 dynamometer was combined with the wake of a vertical plate of 0.85m length and 0.02m thickness which was placed between the trailing edge of the dynamometer strut and the model propellers with a diameter of 250mm, as shown in Figure 3. The wake plate was also covered with a sand paper of grit P36 to trip the wake flow in turbulent regime.

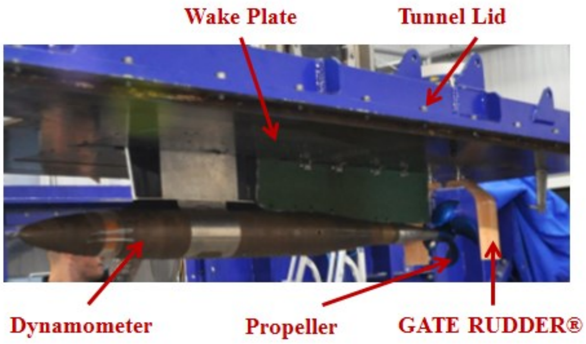

Figure 3. Test set-up with GATE RUDDER® at ECT including wake plate.

During the tests, propeller thrust and torque as well as the shaft rpm were recorded a data collection rate of 100Hz. The URN characteristics were recorded by using a B&K 8103 miniature hydrophone located inside the tunnel in a streamlined strut aligned with the tunnel flow. The cavitation observations were recorded by using moving and still cameras from the side and bottom windows of the ECT for each test condition as well as the oxygen content and temperature of the tunnel water.

3.2 Propeller and Rudder Geometries

The model propeller and rudder geometries for the conventional rudder and GATE RUDDER® propulsion systems were provided by KAMOME Propeller Co, LTD. The same Controllable Pitch Propeller (CPP) model of 250mm diameter with four-blades and high skew was used behind the conventional rudder and GATE RUDDER® systems as shown in Figure 4.

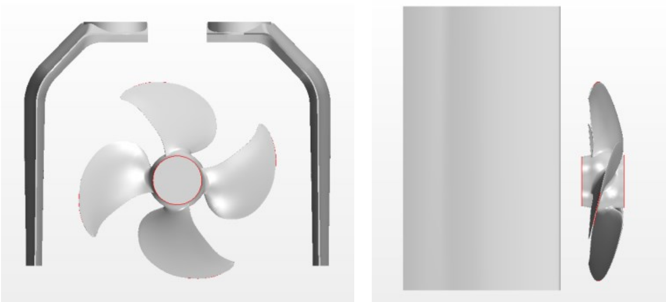

Figure 4. GATE RUDDER® arrangement (Left) and Conventional Rudder arrangement (Right)

3.3 Test Conditions

The cavitation tunnel tests were conducted at 5 different test conditions that represented the equivalent full-scale operational conditions of the container ships. Table 2 presents the test conditions with advance velocity ratio (J), tunnel speed (V), revolution speed (n), tunnel (P_{tun}) and vacuum pressure (P_{vac}) and tunnel temperature (T) parameters that have been set during the tests.

Table 2. Test Conditions

Test Conds’	*J*	*V*	*n*	*Ptun*	*Pvac*	*T*
		m/s	rpm	mmHg	mmHg	°C
Cond’ 1	0.000	0.000	1200	830.7	-200	17.1
Cond’ 2	0.154	0.925	1438	830.7	-200	17.1
Cond’ 3	0.260	1.560	1438	830.7	-200	17.1
Cond’ 4	0.501	3.000	1438	830.7	-200	17.1
Cond’ 5	0.494	3.970	1925	830.7	-400	17.1

The tests were first conducted with the conventional rudder-propeller system arrangement for the above stated conditions and this was followed with the GATE RUDDER® propulsion set-up for the same conditions. During the tests the associated test data for the propeller performances, cavitation observations and URN were collected with each test set-up and analyzed for the comparisons of the data between the conventional rudder propeller and GATE RUDDER® propulsion system as well as to support the CFD studies.

4 COMPUTATIONAL FLUID DYNAMICS (CFD) INVESTIGATIONS

Although 5 different operating conditions were simulated during the cavitation tunnel tests, only one cavitating condition, which produced the strongest tip vortex cavitation (Condition 5), has been presented in the CFD simulations as given in Table 3.

Table 3. EFD and CFD Conditions

Conditions	J	V	n	σ_n
	[-]	[m/s]	[rpm]	[-]
EFD Condition 5	0.494	3.970	1925	1.714
CFD Condition 5	0.500	3.000	1440	1.730

In Table 3 J is the advance velocity ratio (or coefficient) of the propeller given by Equation 3, V is the tunnel in-flow speed, n is the propeller shaft rotational speed and σ_n is the propeller cavitation number based on the shaft speed as described in Equation 2.

4.1 Numerical Method

The CFD simulations for the two propulsion arrangements and for the above described test condition were carried out by using in the well-known commercial CFD software, STAR-CCM+ for marine applications. For the cavitation simulation, two fluids (water and vapour) medium, which are described in the software, and the Volume of Fluid (VOF) method was used for multiphase modelling.

Based on the experience with the rotational fluid domains, for describing the effect of the propeller rotation, the overset mesh method was preferred instead of the sliding mesh approach to be able to simulate the tip vortex cavitation in combination with the rudder and hence to eliminate the data transfer problems between the rotating and stationary domain.

For turbulence modelling, Large Eddy Simulation (LES) turbulence models were preferred for cavitation simulations. In contrast to the Reynolds Average Navier-Stokes (RANS) model, scale-resolving simulations are able to solve the large scales of turbulence and model small-scale motions. For scale-resolving simulations, there are two approaches involving Detached Eddy Simulation (DES) and LES which are available in STAR-CCM+ (STAR-CCM+ User Guide, 2018). LES turbulence model has been preferred more commonly for simulating complex flows such as cavitation, especially for the tip vortex type of cavitation.

The Schnerr-Sauer cavitation model, which is based on Rayleigh-Plesset equation, was also used for this study to simulate the cavitation. The bubble growth rate in the Schnerr-Sauer model (Schnerr & Sauer, 2001) was estimated by using Equation 1 as follows,

$$\left(\frac{dR}{dt}\right)^2 = \frac{2}{3}\left(\frac{p_{sat} - p_\infty}{\rho_l}\right) \quad (1)$$

The cavitation number based on the rotational speed of the propeller shaft is defined as follows.

$$\sigma_n = \frac{p - p_{sat}}{0.5\rho_l (nD)^2} \quad (2)$$

where p is the tunnel pressure, p_{sat} is the saturation pressure of water, ρ_l is the density of the fluid, n is the shaft speed and D is the diameter of the propeller.

The advance velocity ratio can be calculated using Equation 3.

$$J = \frac{V_A}{nD} \quad (3)$$

where V_A is the advance velocity of fluid. Thrust and torque coefficient of the propeller is calculated as follows.

$$K_T = \frac{T}{\rho n^2 D^4} \quad (4)$$

$$K_Q = \frac{Q}{\rho n^2 D^5} \quad (5)$$

where T and Q are thrust and torque values of the propeller respectively and ρ is the density of water. Using K_T and K_Q, the propeller open water efficiency is calculated using Equation 6.

$$\eta_0 = \frac{J}{2\pi}\frac{K_T}{K_Q} \quad (6)$$

4.2 Computational Domain Preparation

As stated earlier, for modelling of the rotation effect, the overset mesh method was used to eliminate the data transfer problems of the sliding mesh approach that may occur between the rotating and stationary domains during the stretching tip vortices from the tip of the propeller blades through the rudder geometry. Within the scope of this study, two different flow domains were prepared for the cavitation simulations, which are associated with the conventional rudder and the GATE RUDDER® configurations. Accordingly, two regions were prepared as the background and overset regions for the simulations of both propulsion systems.

Figure 5 presents the flow domain has been prepared for cavitation simulations of the GATE RUDDER® system including the background and overset mesh regions.

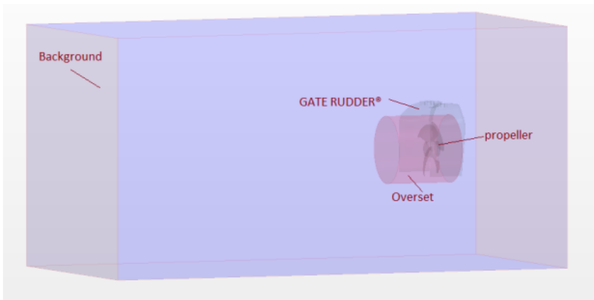

Figure 5. Computational Flow Domain for GATE RUDDER® system

4.3 Mesh Generation

4.3.1 Sheet Cavitation

A suitable mesh arrangement was generated for each computational case for the sheet cavitation simulations on the propeller blades. While a 0.006D surface size for the mesh generation was applied on the propeller surfaces in general, smaller surface size with a 0.004D was preferred for a volumetric control around the propeller tip regions with a cylinder geometry.

Figure 6 presents the generated mesh for sheet cavitation simulations for the conventional rudder-propeller and GATE RUDDER® propulsion systems, respectively.

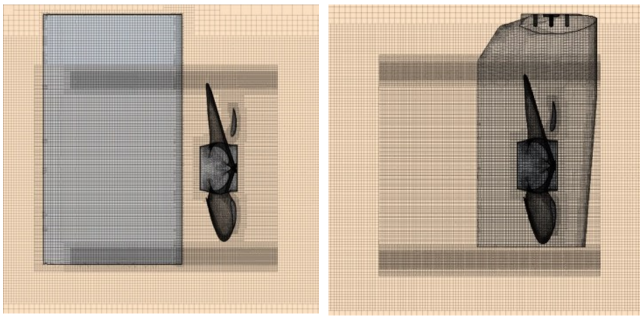

Figure 6. Generated mesh for sheet cavitation

Conventional Rudder-propeller system (Left); GATE RUDDER® Propulsion system (Right)

4.3.2 Tip Vortex Cavitation

Although the sheet cavitation could have been simulated successfully using the mesh arrangement, which is shown in Figure 6, it was expected that the existing mesh and analysis methods were not sufficient to capture the tip vortex cavitation, and to predict the propeller performance accurately, as reported in the open literature [e.g. Viitanen & Siikonen, 2017, Lloyd et al., 2017, Shin & Anderson, 2018]

For capturing a sudden pressure drop and cavity bubbles in a propeller slipstream, an adaptive mesh refinement approach has been developed by the leading author of the present paper. The new mesh refinement approach, which is called MARCS (Mesh Adaption Refinement for Cavitation Simulations), has been presented using various standard test propellers such as INSEAN E779A, PPTC and the Princess Royal propellers in the past (e.g. Yilmaz et al, 2018). This method was also applied in this study to simulate the tip vortex cavitation for the both propulsion systems.

In the MARCS procedure, the mesh was refined only in the region where the tip vortex cavitation may occur in propeller slipstream. Before the application of this procedure, the simulation was run and sheet cavitation was simulated using the coarse mesh arrangement as shown in Figure 6. At the end of this simulation, using the existing solution, the q-criterion limit was determined by creating a threshold region in the STAR-CCM+ software as shown in Figure 7 (on Left).

In the cavitation simulations, the volume fraction of the vapour indicates the regions of the cavity volume where the absolute pressure drops below the saturated vapour pressure of the water, thus demonstrates the cavitating volume. In the meantime, a region was prepared by using the q-criterion to define the zone where the vortices have been developed, thus generating the blue region as shown in Figure 7 (Left). The combination of the both regions provides a specification of the volumetric trajectory on which an adaptive mesh generation mechanism for capturing the sudden pressure drop region and tracking the tip vortices in the propeller slipstream rather effectively and accurately.

Within the framework of the MARCS approach, a field function was created to generate finer meshes where the q-criterion was above 20000s^{-2}. Having generated the finer meshes, a mesh refinement table, which included the coordinates of each cell needed to be refined and their surface sizes, was prepared automatically by STAR-CCM+ using the suitable field functions to generate meshes. Figure 7 presents the isosurface of the q-criterion above 20000s^{-2} (Left) and generated mesh (Right) using the refinement table that was prepared by using the q-criterion trajectory.

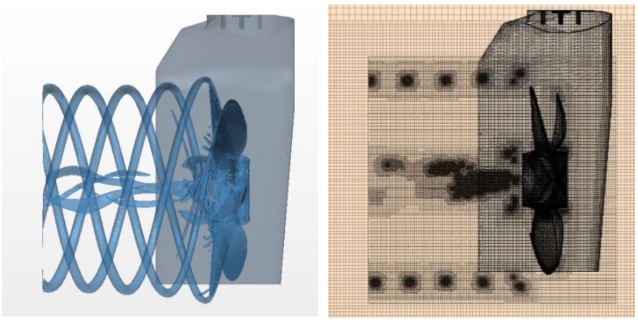

Figure 7. Generated Mesh for Tip Vortex Cavitation

(Left; Q-Criterion > 20000s^{-2}, Right; Mesh)

In order to familiarize the reader with the further details of the applied MARCS approach, a flowchart is provided in Figure 8 to demonstrate the sequential steps of this approach.

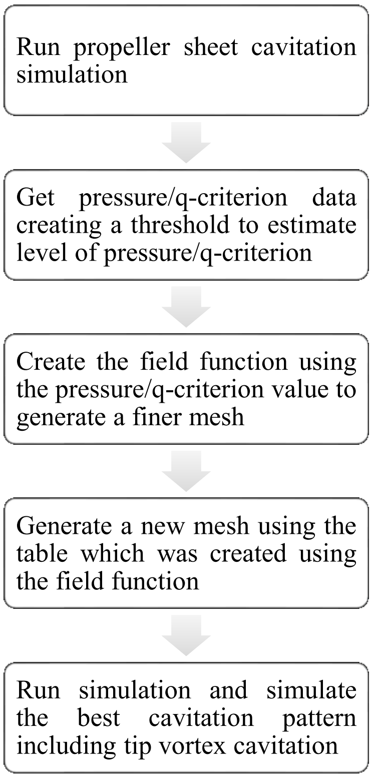

Figure 8. Flow chart summarizing MARCS approach

5 RESULTS AND DISCUSSION

As stated earlier, although 5 different operating conditions were simulated during the cavitation tunnel tests, which are shown in Table 2, only one cavitating condition, which produced the strongest tip vortex cavitation (Condition 5), was simulated in the CFD simulations as given in Table 3, for the both propulsion systems.

Table 4 shows the comparative propeller performance characteristics (*ie* K_T, K_Q *and* η_o) of the both propulsion systems based on the CFD simulations (analysis) and cavitation tunnel test measurements (i.e. EFD analysis).

Table 4. EFD and CFD Results Comparisons between conventional and GATE RUDDER®

Conditions		K_T	$10K_Q$	η_0
		[-]	[m/s]	[rpm]
Conventional Rudder	EFD	0.2156	0.2910	0.5835
	CFD	0.2071	0.2717	0.6067
	Deviation	3.9%	6.6%	-4.0%
GATE RUDDER®	EFD	0.1716	0.2497	0.5415
	CFD	0.1712	0.2374	0.5741
	Deviation	0.2%	4.9%	-6.0%

As shown in Table 4, although the CFD predictions for the performance of the propeller shows a good agreement with the experiments (EFD), especially for the GATE RUDDER® system simulations in terms of K_T, for which the deviation is less than 1%, K_Q could only be predicted within a %5 and %6.6 deviation for the GATE RUDDER® and the conventional rudder system, respectively. The deviation in the K_Q predictions can be related to the geometrical differences, the presence of the wake plate and the similar (but not exact) conditions between the EFD and CFD predictions due to the time restrictions of this study that requires further fine tuning and investigations.

As far as the CFD predicted cavitation patterns are concerned, Figure 9a and Figure 9b shows the sheet cavitation and tip vortex cavitation, respectively, in comparison for the conventional rudder-propeller system (Left) and GATE RUDDER® propulsion system (Right).

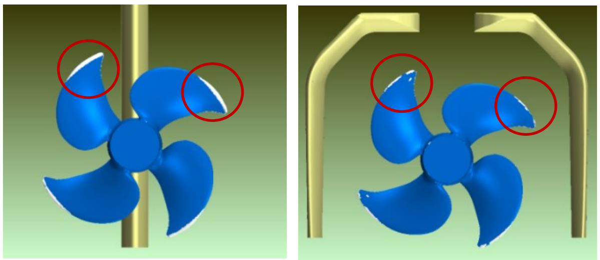

Figure 9a. Sheet Cavitation Comparisons (CFD) Conventional Rudder (Left); GATE RUDDER® (Right)

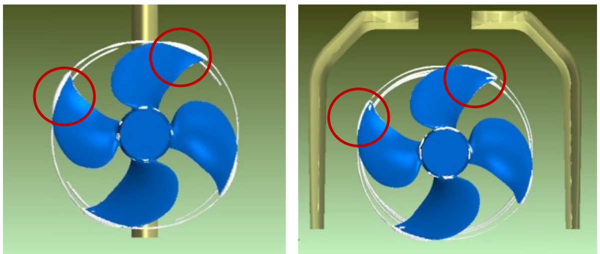

Figure 9b. Tip Vortex Cavitation Comparisons (CFD) Conventional Rudder (Left); GATE RUDDER® (Right)

As clearly shown in Figure 9a and 9b, not only the sheet vortex extent and volume on the blades but also the tip vortex volume and strength was reduced on the GATE RUDDER® system (Right) compared to the conventional rudder-propeller system (Left). The reduction on the cavitation volumes associated with the GATE RUDDER® arrangement is also the indication for the reduced propeller induced vibrations and URN levels compared to the conventional rudder-propeller system.

The comparative cavitation patterns for the conventional rudder-propeller system and GATE RUDDER® propulsion system, as observed from the cavitation tunnel tests (Left) and from the CFD simulations (Right), are presented in Figure 10a and 10b, respectively.

As observed during the cavitation tests with the conventional rudder-propeller system, a strong sheet cavity was covering almost a 20% of each blade surface and more accentuated at the top dead center (i.e. wake shadow region), as shown in Figure 10a (Left). Due to the effect of the wake plate, the deformation of the tip vortices at the same region was also observed and this deformation was combined with the effect of the rudder in downstream resulting in the bifurcation of the tip vortex at the rudder leading edge. In spite of the accentuated sheet cavity dynamics at the wake shadow and deformation of the tip vortex at the rudder leading edge, the tip vortex cavitation was transported in downstream through the propeller slipstream and the rudder without losing its strength.

The above described cavitation pattern and part of the cavity dynamics can be also observed in the CFD simulations when once compares the left and right illustrations in Figure 10. The dynamics resulting from the wake plate will not be reflected on the results due to the time restrictions.

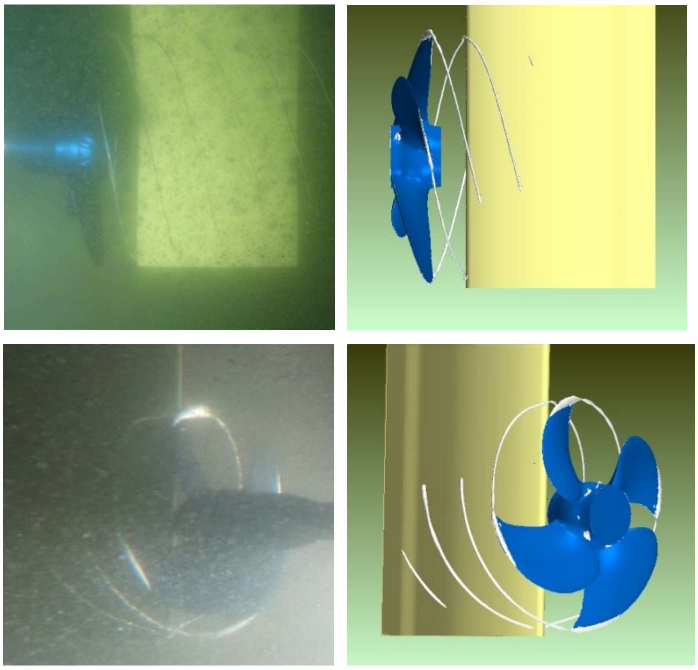

Figure 10a. Cavitation Comparisons for Conventional Rudder-propeller system

(Left: EFD from tunnel tests; Right; CFD predictions)

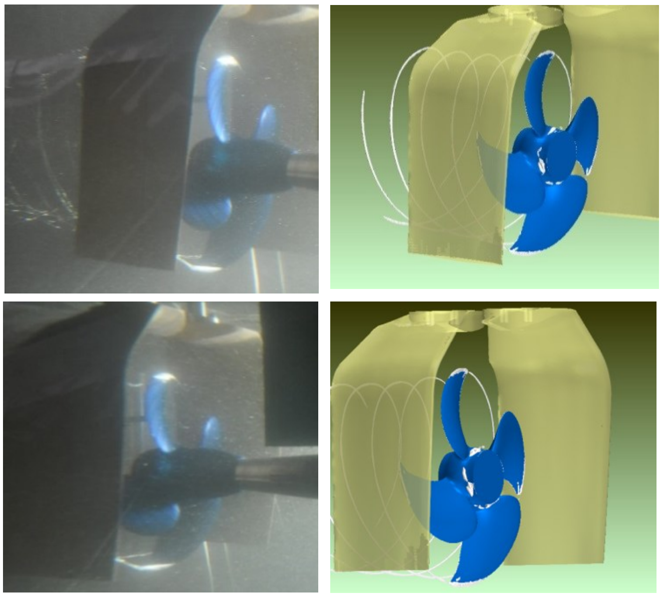

Figure 10b. Cavitation Comparisons for GATE RUDDER® Propulsion system

(Left: EFD from tunnel tests; Right; CFD predictions)

On the other hand, the cavitation observations with the GATE RUDDER® arrangement in the tunnel indicated that the sheet cavitation on the blades was developed at a lesser extent, about 15% of each blade, in comparison to the conventional rudder case, and it was combined with a reduced strength of the tip vortex cavitation as shown in Figure 10b. In contrast to the observations with the conventional rudder-propeller arrangement, the tip vortex cavitation developed on the GATE RUDDER® propeller had no deformation or bifurcation, as expected, extending smoothly in the downstream at a reduced strength. These patterns and cavity dynamics were also captured well with the CFD simulations, thanks to recently developed MARCS procedure.

6 CONCLUDING REMARKS

The investigations on the cavitation performance of a newly introduced novel ESD, the GATE RUDDER®, was conducted by using the CFD and EFD approaches in comparison with the cavitation performance of a conventional rudder-propeller system. The investigation aimed to shed a light on the reduced hull vibrations and quieter aft end performance experienced with the world's first GATE RUDDER® system fitted on a container vessel compared to its sister ship with the conventional rudder-propeller system. The investigation also aimed to explore the cavitation performance of this novel ESD by using a state-of-the-art CFD tool and associated MARCS procedure validated by the cavitation tunnel tests for the first time. The investigations conducted so far have indicated that:

- The model test data and supporting CFD predictions are the first information reported to on the cavitation performance of the GATE RUDDER® system in comparison with that of a conventional rudder-propeller system.

- Yilmaz's recently developed new adaptive mesh refinement technique (MARCS) successfully captured the cavitation performance characteristics of the GATE RUDDER® as well as the conventional-rudder, especially with the interaction of the tip vortices with the rudder arrangements, based on the comparison with the EFD results.

- Based on the EFD and CFD investigations conducted in the model-scale with a relatively simple hull wake simulation arrangements, the GATE RUDDER® propulsion system can display reduced sheet and tip vortex volumes and variations compared to those of the conventional rudder-propeller arrangement. No observation was made with the GATE RUDDER® for the deformed and hence bifurcated tip vortex at the rudder leading edge of the conventional rudder and hence lesser cavity dynamics.

- Although the above findings are based on the model-scale investigations with relatively simple hull wake arrangements, they may strongly support the lesser vibrations and quieter aft end characteristics of the GATE RUDDER® experienced onboard by the ship crew.

There is no doubt that the results in this paper presents only the preliminary GATE RUDDER® investigations and hence require further work regarding: i) More detailed CFD modelling of the current test case; ii) More sophisticated or representative modeling of the model ship arrangements, preferably using a full hull model in a larger test facility; iii) Further CFD simulations at the full-scale. Regarding further work (i) we will be improving our CFD model in terms of the propeller hydrodynamic performance and cavitation patterns by using the exact tunnel details by including the wake plate arrangement that could not be included due to the time restrictions of this paper, as shown in Figure 11.

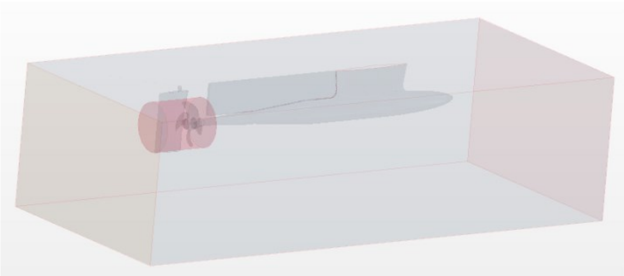

Figure 11. Flow domain for the new CFD simulations including dynamometer and wake plate geometries

ACKNOWLEDGEMENTS

The principal author of this paper is sponsored by the Turkish Ministry of Education during this study. The access provided to High Performance Computing for the West of Scotland (Archie-West) thorough EPSRC grant no. EP/K000586/1 is gratefully acknowledged. The authors are also grateful to KAMOME Propeller Co, Ltd. for providing the model propeller and rudders as well as supporting the tunnel experiments at ECT.

REFERENCES

Atlar, M. (2011). "Recent upgrading of marine testing facilities at Newcastle University". In: Proceedings of the 2nd International Conference on Advanced Model Measurement Technologies for the Maritime Industry (AMT'11). Newcastle upon Tyne, UK.

Carlton, J. (2012). "Marine propellers and propulsion" 3rd Edition, 2012.

Lloyd, T., Vaz, G., Rijpkema, D., Reverberi, A., (2017), Computational fluid dynamics prediction of marine propeller cavitation including solution verification, Fifth International Symposium on Marine Propulsors smp'17, Espoo, June 2017.

Sasaki, N., Atlar, M., and Kuribayashi, S. (2015) "Advantage of twin rudder system with asymmetric wing section aside a propeller" Journal of Marine Science and Technology.

Sasaki, N., Kuribayashi, S., Atlar, M. (2018). "GATE RUDDER ®", 3rd International Symposium on Naval Architecture and Maritime, 24-25 April, Istanbul, Turkey.

Schnerr, G. H., Sauer, J. (2001). "Physical and Numerical Modeling of Unsteady Cavitation Dynamics", ICMF-2001, 4th International Conference on Multiphase Flow, New Orleans, USA, May 27-June 1, 2001

Shin, W. K., Andersen, P., (2018), "CFD Analysis of Propeller Tip Vortex Cavitation in Ship Wake Fields", The 10th International Symposium on Cavitation (CAV2018), Baltimore, Maryland, USA, 14-16 May 2018.

STAR-CCM+ User Guide, 2018

Viitanen, V, M, Siikonen, T., (2017), Numerical Simulation of Cavitating Marine Propeller Flows, 9th National Conference on Computational Mechanics, MekIT'17.

Yilmaz, N., Atlar, M., Fitzsimmons, A., P. (2018). "An Improved Tip Vortex Cavitation Model for Propeller-Rudder Interaction", The 10th International Symposium on Cavitation (CAV2018), Baltimore, Maryland, USA, 14-16 May 2018.