*11$^{th}$ International Workshop on Ship and Marine Hydrodynamic*
*Hamburg, Germany, September 22-25, 2019*

# Different Techniques To Simulate Tandem Propeller with Boundary Element Method

**Youjiang Wang**[1,*], **Ulf Göttsche**[1], **Moustafa Abdel-Maksoud**[1], **Stefan Krüger**[2]

[1]Hamburg University of Technology, Institute for Fluid Dynamics and Ship Theory
Am Schwarzenberg-Campus 4(C), 21073 Hamburg, Germany
[2]Hamburg University of Technology, Institute of Ship Design and Ship Safety
Am Schwarzenberg-Campus 4(C), 21073 Hamburg, Germany

[*]Corresponding author, youjiang.wang@tuhh.de

## ABSTRACT

In this work, the boundary element method (BEM) has been used to calculate the open water performance of a tandem propeller with our in-house solver ***pan*MARE** . Simulating such multi-stage marine propulsors involves handling the interaction between the rear blades and the wake sheets of front blades. The wake sheet and blade geometry can intersect, which may lead to singularity and incorrect pressure distribution. Different techniques are proposed to allow a simulation without singularity. The results obtained with these techniques are presented, and their advantages and disadvantages are discussed.

## 1 INTRODUCTION

The boundary element method (BEM) is widely used for marine propeller design and analysis. BEM is based on potential theory and helps predict the propeller performances efficiently. It is especially useful to assess and optimize designs in the early stage.

In BEM, the vorticity shedding from the trailing edge is modelled mathematically by a vortex sheet (or wake sheet). In practice, sometimes the multi-stage marine propulsors (tandem propeller, rotor-stator configuration, contra-rotating propeller) need to be calculated. Simulating multi-stage marine propulsors involves handling the interaction between the rear blades and the wake sheets of front blades. The wake sheet and blade geometry can intersect, and this may lead to singularity and incorrect pressure distribution. In [1], this problem was solved by combining the lifting line method and BEM. In current work, we try to solve it only in the framework of BEM.

A tandem propeller, which was described in [1, 2], is to be analysed with BEM. Different techniques are proposed to avoid the singularity. The obtained results are presented and compared. Their advantages and disadvantages including the applicability for other types of multi-stage marine propulsors are discussed. Our in-house BEM code ***pan*MARE** was used for the calculation. ***pan*MARE** has been successfully applied in many fields, including the prediction of propeller performances in open water or a ship wake field, cavitation, pressure fluctuation, wave-body interaction, etc.[3–7].

The manuscript is organized as follows. In the next section the BEM theory is briefly described. Then different techniques for the simulation of tandem propellers are presented in section 3. The numerical results are given in section 4. Finally the conclusion and discussion are presented in section 5.

## 2 BOUNDARY ELEMENT METHOD

Here, only a brief introduction is presented to allow a better understanding of the following sections. More details about the boundary element method can be found in [8–10]. Boundary element method is based on the inviscid potential theory. The disturbed velocity potential $\phi$ is the main unknown. To get the distribution of $\phi$ in the fluid domain, only its value and normal derivative on the boundaries are necessary, which are mathematically represented by dipole and source, respectively. The boundaries for potential flow domain include the solid body boundary as well as a wake sheet representing the vorticity slip stream, as shown in Fig. 1. Physically there should be no normal velocity on the wake sheet. Based on this requirement the wake sheet geometry is iteratively determined during the calculation, which is called the wake alignment.

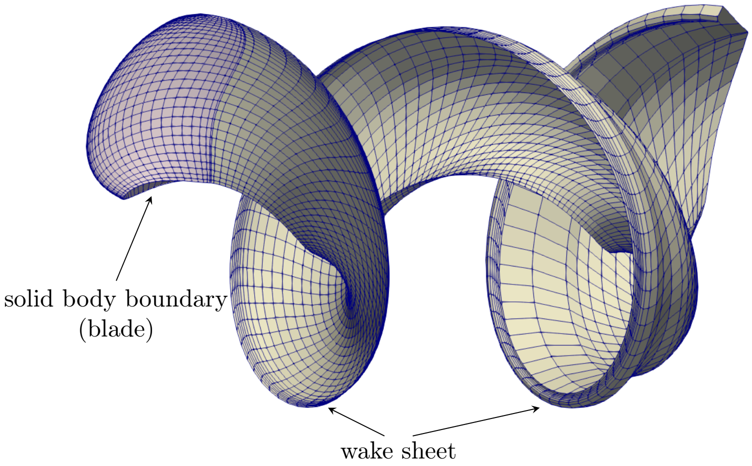


Figure 1: Mathematical boundaries in BEM. Only one blade is showed.

The boundaries are discretized into panels (i.e. boundary elements) for numerical solution. On every panel, there are source and dipole with constant strengths (on wake panel the source strength is always zero). The source and dipole induce potential everywhere in the fluid domain. With boundary conditions on selected points (normally the body panel centers), the source and dipole strengths can be obtained by solving linear equation system. In current work, the sources are known from the inflow velocity, and dipoles are unknown and solved using the Dirichlet boundary condition. For more details please refer to the above mentioned citations.

With potential, the fluid velocity can be calculated. Then the pressure distribution is obtained according to Bernoulli equation. Viscous frictional force is also considered by using empirical corrections. The thrust and torque of the propeller are finally obtained by integrating the pressure and viscous forces.

## 3 TECHNIQUES TO SIMULATE TANDEM PROPELLER

In the tandem propeller, two co-rotating components are fixed on the same hub one after the other. The rear propeller works in the slip stream of the front propeller. When BEM is used directly for the analysis of tandem propeller, the wake panel may get very close or even intersect with the body panel, as shown in Fig. 2. This leads to nonphysical singularity during solving dipole strength or calculating pressure. The situation is similar for other types of multi-stage marine propulsors.

In this section, different techniques to solve this problem are described. They are all easy to realize thanks to the various functions in our in-house code ***pan*MARE** .

### 3.1 Panel deactivation

Just as its name implies, with this technique a part of wake panels will be deactivated ( or rather mathematically deleted). Different from the other techniques, the two propeller components are simultaneously simulated in the same solver.

During the wake alignment procedure, the points of the wake sheet are kept at a specific distance from the blade surface. It can be considered as that there is a virtual fence around the blade surface. The streamwise edges on the wake sheet are not allowed to intersect with this fence (i.e. wake sheet points can not go into the fence). If it happens, the downstream point is moved out in the normal direction and placed on the fence surface.

Under the above mentioned conditions, wake panels may still intersect with the blade geometry, as shown in Fig. 2. In such cases they are removed from the solver (deactivated) together with all downstream panels in the same streamwise strip, as the light red panels showed in Fig. 2. Vortices with large circulation are formed on the edge of the removed panels. They mimic the effects of the stretched vortex near the blade body. Downstream of the rear propeller, these vortices rotate around each other, and cancel out each others' effect.

With this technique, a specified distance is always kept between the blade panel and wake panel. Therefore, singularity is avoided.

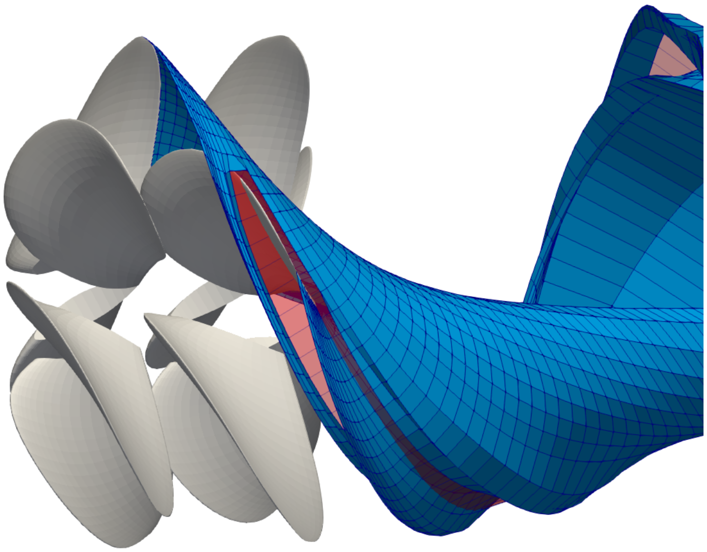


Figure 2: Example of wake panel deactivation technique. Gray stands for blades, blue for active wake panels, and light red for inactive wake panels.

### 3.2 Multi solver method

Another approach is to use separate solvers for each propeller component, which can be called MS (Multi Solver) technique. In each solver only one single propeller is simulated. Mutual induced velocities on the propeller plane are used to consider the interaction effects, as shown in Figure 3. The mutual induced velocity is extracted from one solver and applied to the other solver. In the following text, the mutual induced velocity is called the interaction velocity, and denoted by $\Delta V_a$, $\Delta V_t$ and $\Delta V_r$ for its axial, tangential and radial components respectively.

Instead of using the extracted interaction velocity field directly, it can be firstly averaged to simplify the simulation. Different averaging levels result in various techniques. They include :

- **MS-NA**: No Averaging MS, the spatially varying interaction velocities are directly used as the virtual wake field.
- **MS-CA**: Circumferential Averaging MS, the interaction velocities are firstly averaged in the circumferential direction before use.
- **MS-PA**: Plane Averaging MS, the interaction velocities are averaged on the whole propeller plane.

In Fig. 4 the typical interaction velocity fields obtained with different techniques are shown.

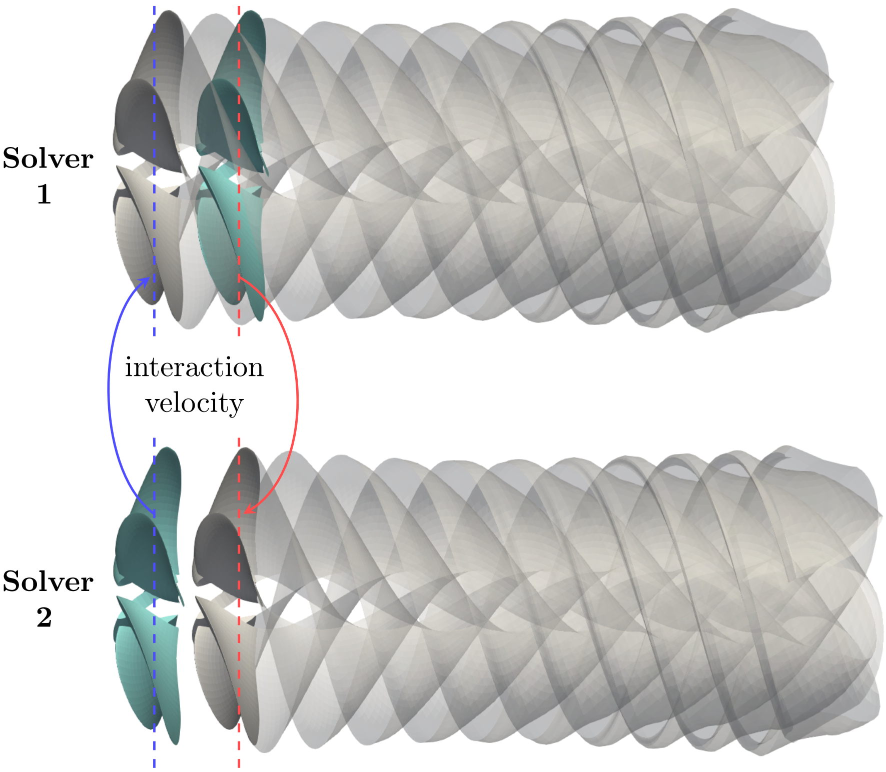
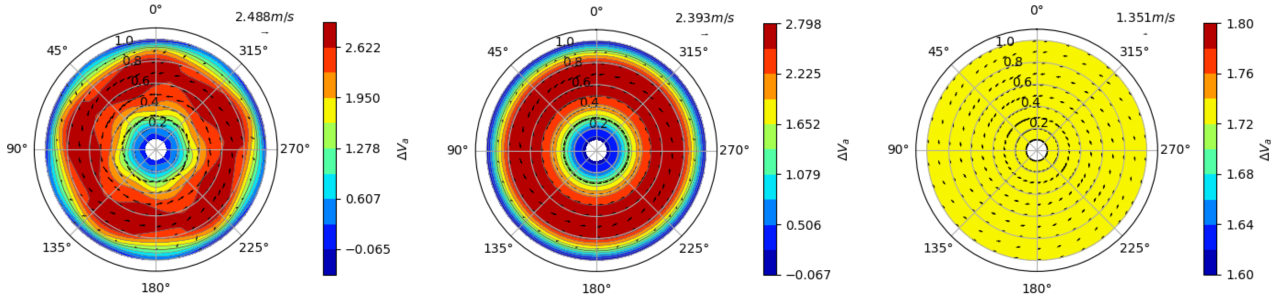


Figure 3: Multi solver technique and interaction velocities. (The propeller with light-blue color with not be simulated in corresponding solver.)



(a)MS-NA (b)MS-CA (c)MS-PA

Figure 4: The interaction velocity applied to the rear propeller with different techniques. The condition is at $J = 0.8$ and it is seen from upstream.

## 4 NUMERICAL RESULTS

The being simulated propeller is the EPROSYS tandem propeller given in [2]. It consists of two components, i.e. EPROSYS 2764 in the front and EPROSYS 2766 in the rear. They have both 5 blades, and the front component has a larger diameter than the rear one.

Before calculating the tandem propeller, each component is simulated alone for validation. Rotational symmetry condition is used, and on the blade surface $24 \times 28$ (chord $\times$ span) panels are used. The obtained open water performances are given in Fig. 5 together with the measured data. The calculated results correlate generally well with the experimental measurements. For small advance ratios, deviations can still be observed for the torque coefficient ($K_Q$). This is a common phenomenon for Potential code.

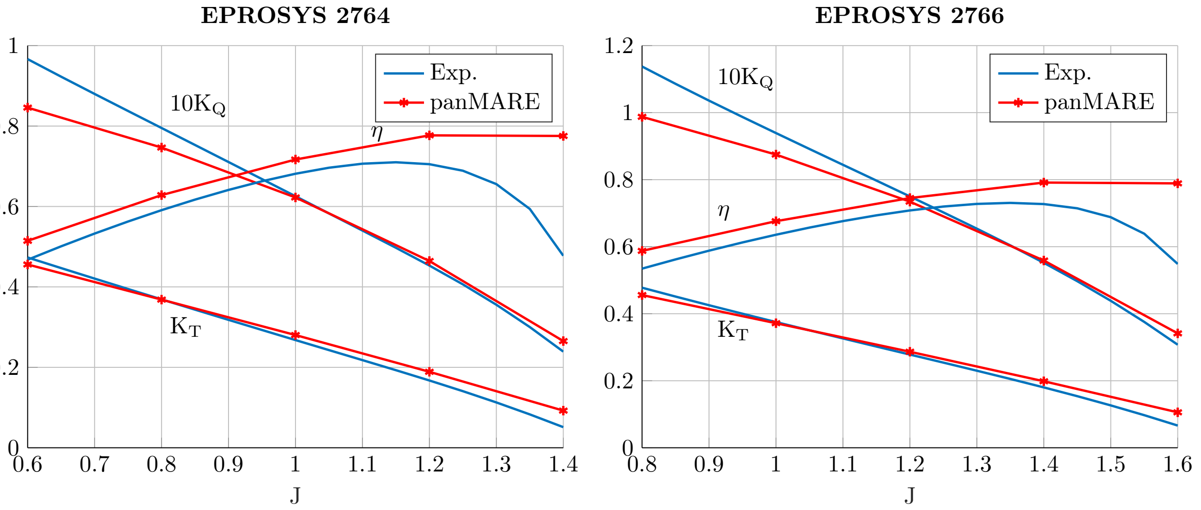


Figure 5: Open water characteristics of the single propeller component.

Then the different techniques described in section 3 are used to simulate the tandem propeller. The obtained open water characteristics are shown in Fig. 6. The differences between the results obtained with different techniques are obvious. MS-NA and MS-CA produce the similar results. Panel deactivation leads to a little smaller $K_T$ and $K_Q$ than them while MS-PA gives much larger values. Compared with the measured values, MS-PA predicts the torque coefficient $K_Q$ quite well and other methods predict the thrust coefficient better.

To understand the differences, the load on both propeller components are given in Fig. 7. The thrust and torque on the front component obtained with different techniques are almost the same, and the differences appear mainly on the rear component. The rear component works in the slip stream of the front one, is therefore more sensitive to the method used ot handle the interaction. The interaction velocity applied to the rear propeller with MS-CA and MS-PA techniques are shown in Fig. 8. With MS-CA, the axial and tangential velocities both achieve maximum in the radius range $[0.4R, 0.8R]$, and decrease when $r$ is outside of this range. Because of radial averaging, MS-PA does not hold such a variation, and the velocities are the same for all positions. Such a difference leads to different effective working conditions of the rear propeller. Here we define a local effective advance ratio as

$$J_{eff}(r) = \frac{V_a + \Delta V_a}{(n - \Delta V_t/(2\pi r))D} \tag{1}$$

where $V_a$ is the undisturbed inflow velocity for the tandem propeller, $n$ is the rotation rate and $D$ is the rear propeller diameter. The distribution of $J_{eff}$ on the rear propeller at $J = 0.8$ with $J = V_a/nD$ are given in Fig. 9. With MS-PA, most of the blade regions work at a smaller effective advance ratios than with MS-CA, and therefore larger thrust and torque are predicted.

In Fig. 8 it can also be observed that the rear propeller produces quite small thrust and torque, irrespective of the $J$ value. The reason is that the effective advance ratio $J_{eff}$ of the rear propeller

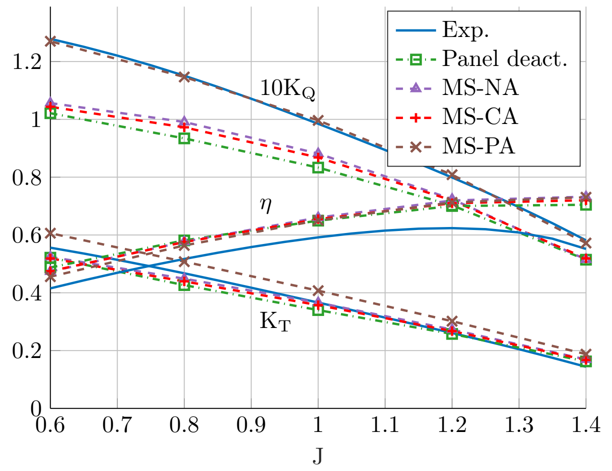


Figure 6: Open water performances of the tandem propeller.

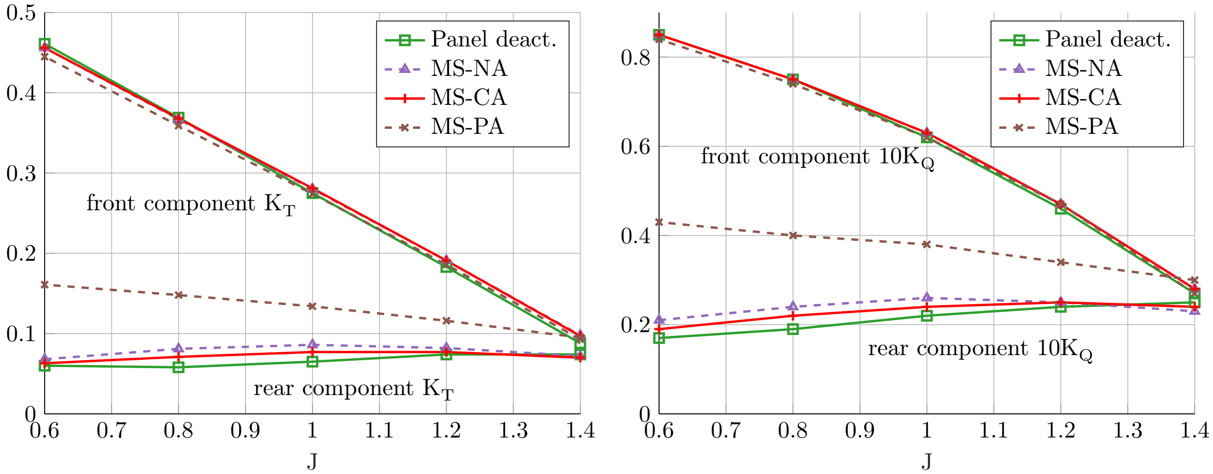


Figure 7: Load distribution between the front and rear propeller components.

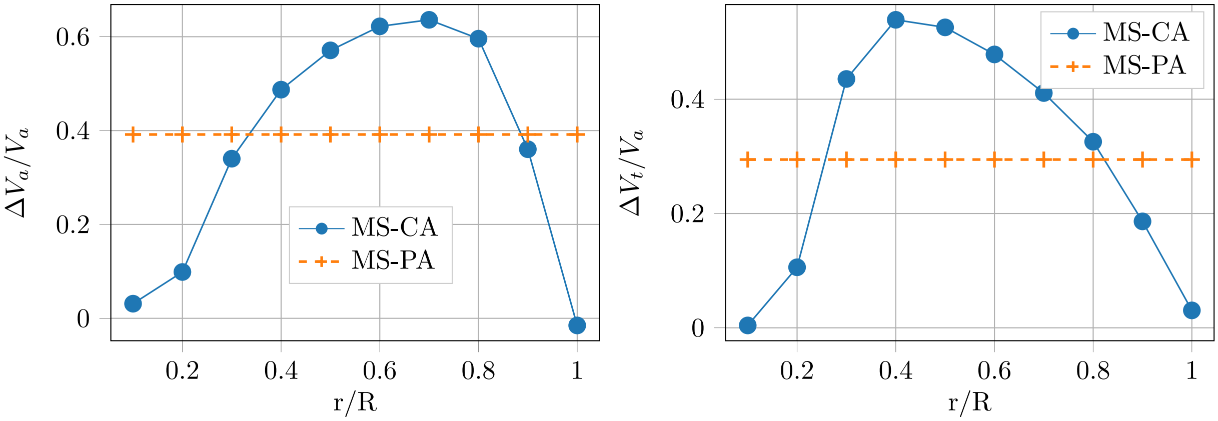


Figure 8: Applied interaction velocities on the rear propeller component with different techniques.

does not change significantly, as shown in Fig. 10. The interaction effect makes it always work around $J_{eff} = 1.65$. Thus, to achieve better performances, the rear propeller should be designed respecting this condition.

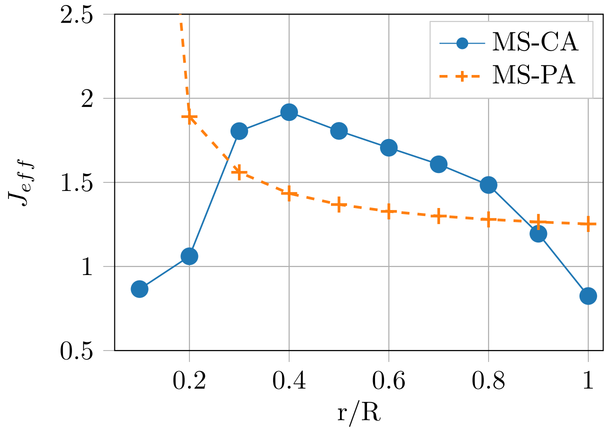


Figure 9: Distribution of the effective J.

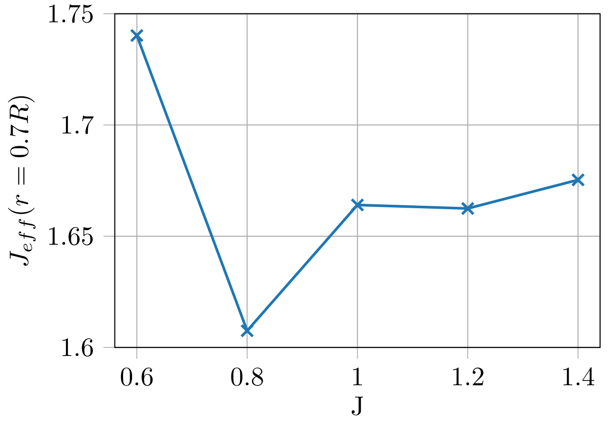


Figure 10: Variation of the rear component's effective J with nominal J.

## 5 DISCUSSION AND CONCLUSION

Different numerical techniques to simulate multi-stage marine propulsor with boundary element method are described and applied for the analysis of a tandem propeller. They include the panel deactivation, multi-solver without velocity averaging (MS-NA), multi-solver with circumferential velocity averaging (MS-CA) and multi-solver with plane velocity averaging (MS-PA). Panel deactivation, MS-NA and MS-CA techniques lead to similar results. The obtained thrust is satisfactory while the torque is too small compared to the experimental data. With MS-PA technique, the torque is predicted much better but the thrust is too large. It is also observed that the rear propeller works always at quite high effective advance ratios irrespective of the advance ratio of the whole tandem propeller. Such effect should be considered during the propeller design.

Based on this study, we propose to chose MS-CA technique for further simulation of multi-stage marine propulsor. Although MS-PA predicts the torque better than other techniques in current work, it can be misleading during the propeller design, because it provides no information about radial variation of the velocity. In MS-CA such informations are preserved. MS-NA leads to similar results as MS-CA, but requires many unsteady iterations for other types of multi-stage propulsors (e.g. contra-rotating propeller). Panel deactivation technique is difficult to implement, and does not give more benefits. As a conclusion MS-CA is the best choice for multi-stage marine propulsor simulation.

Further work includes more validation studies and understanding the discrepancy between the experimental data and the predicted results.

## ACKNOWLEDGEMENTS

The authors would like to thank the German Federal Ministry of Economic Affairs and Energy (BMWi) for its support in the frame of the project ProAkus (part of the European project ProNoVi).

## REFERENCES

[1] S. Krüger, Y. Wang, S. Martin, U. Göttsche and M. Abdel-Maksoud. “A hybrid calculation concept for single and multi-component propulsors”. In: *Sixth International Symposium on Marine Propulsors (SMP’19).* Rom, Italian, 2019.

[2] C. Johannsen. *Final report of the research project EPROSYS.* Tech. rep. 1662. Hamburg, Germany: HSVA, 2007.

[3] M. Greve and M. Abdel-Maksoud. “Potential theory based simulations of unsteady propeller forces including free surface effects”. In: *Fourth International Symposium on Marine Propulsors (SMP’15).* Austin, Texas, USA, 2015.

[4] S. Berger, M. Scharf, U. Göttsche, J. Neitzel, R. Angerbauer and M. Abdel-Maksoud. “Numerical Simulation of Propeller-Rudder Interaction for Non-Cavitating and Cavitating Flows Using Different Approaches”. In: *Fourth International Symposium on Marine Propulsors (SMP’15).* Austin, Texas, USA, 2015.

[5] U. Göttsche, M. Scharf, S. Berger and M. Abdel-Maksoud. “A Hybrid Numerical Method for Investigating Underwater Sound Propagation of Cavitating Propellers”. In: *Fifth International Symposium on Marine Propulsors (SMP’17).* Espoo, Finland, 2017.

[6] D. F. González, J. Bechthold and M. Abdel-Maksoud. “Application of a Boundary Element Method for Wave-Body Interaction Problems Considering the Non-Linear Water Surface”. In: *ASME 2017 36th International Conference on Ocean, Offshore and Arctic Engineering.* American Society of Mechanical Engineers. 2017, V07AT06A057–V07AT06A057.

[7] S. Netzband, C. W. Schulz, U. Göttsche, D. Ferreira González and M. Abdel-Maksoud. “A panel method for floating offshore wind turbine simulations with fully integrated aero-and hydrodynamic modelling in time domain”. In: *Ship Technology Research* 65.3 (2018), pp. 123–136.

[8] J. Katz and A. Plotkin. *Low-speed aerodynamics.* Vol. 13. Cambridge Aerospace Series. Cambridge, United Kingdom: Cambridge University Press, 2001.

[9] Y. Wang, M. Abdel-Maksoud and B. Song. “Convergence of different wake alignment methods in a panel code for steady-state flows”. In: *JOURNAL OF MARINE SCIENCE AND TECHNOLOGY* 21.4 (2016), pp. 567–578.

[10] Y. Wang, M. Abdel-Maksoud and B. Song. “A fast method to realize the pressure Kutta condition in boundary element method for lifting bodies”. In: *OCEAN ENGINEERING* 130 (2017), pp. 398–406.