Volume XXIX 2026
MBNA Publishing House Constanta 2026

Scientific Bulletin of Naval Academy

SBNA PAPER • OPEN ACCESS

Evaluation of hydrodynamic noise on military ships

To cite this article: Ali Beazit, Sporiș Adriana, Ali Levent, Scientific Bulletin of Naval Academy, Vol. XXIX 2026, pg. 216-222.

Submitted: 10.04.2026
Revised: 15.08.2026
Accepted: 08.09.2026

Available online at www.anmb.ro

ISSN: 2392-8956; ISSN-L: 1454-864X

doi: 10.21279/1454-864X-26-I1-021

Evaluation of hydrodynamic noise on military ships

Beazit Ali[1] *, Adriana Sporiş[2], Levent Ali[3]
[1]"Mircea cel Bătran" Naval Academy, Marine Engineering Faculty, Naval Electromechanical Systems Department *email: ali.beazit@yahoo.com
[2] "Mircea cel Bătran" Naval Academy, Marine Engineering Faculty, Naval Electromechanical Systems Department
[3] "Mircea cel Bătran" Naval Academy, Marine Engineering Faculty, Naval Electromechanical Systems Department

Abstract. The reduction of under-water noise is one of the most difficult tasks facing the naval architect of military ships. Specific fluid dynamics software has become an important tool for supporting the design of military ships, not only with respect to the minimisation of the resistence of the hull, but also with respect to the optimisation of the ship propultion and the reduction of under-water signatures. Computational fluid dynamics (CFD) simulations are employed to analyze velocity and pressure fields, wake characteristics, and vortex structures under full-scale conditions. The results highlight the influence of wake non-uniformity, appendage configuration, and propeller geometry on acoustic performance. Based on the analysis, design recommendations are proposed for minimizing noise through flow optimization and improved propulsive efficiency.
Keywords: pressure distribution, propeller design, velocity distribution, underwater noise, wake field

1.Introduction

Underwater radiated noise generated by marine vessels is an important issue in naval hydrodynamics particularity for vessels for which acoustic signature is directly related to operational performance and detectability. Among the different sources of underwater noise, propeller-induced noise is strongly influenced by the hydrodynamic interaction between the hull, wake field, propeller and appendages.

Previous studies have demonstrated that the non - uniformity of the propeller inflow, cavitation and unsteady propeller loading are important mechanisms contributing to underwater radiated noise. Numerical approaches based on CFD have therefore been increasingly employed to investigate the hydrodynamic and acoustic behavior of marine propeller and ship-propeller configuration (Sakamoto & Kamiirisa et al., 2018); Kalikatzarakis et al., 2022)

Scale effects must also be considered when evaluating the acoustic behaviour of a naval propeller. In particular, the pressure reduction within the tip-vortex core and associated cavitation behaviour can differ significantly between model-scale and full-scale conditions. Consequently, full-scale CFD calculations and appopriate scaling procedures are important for obtaining reliable estimates of proppeler-induced noise. Recent experimental and numerical investigations have shown that full-scale- predictions remain challenging and that model to full scale extrapolation can introduce significant uncertainty, especially when cavitation is present (Lee et al., 2024).

More recent studies have combined CFD - based flow simulations with hydroacoustic approaces to estimate propeller-induced noise, while experimental investigations have highlighted the difficulties associated with transferring model-scale results to full-scale operating conditions (McIntyre et al., 2022).

Nevertheless, the prediction and reduction of underwater noise remain challenging because of the strong dependence of the acoustic response on the spatial non-uniformity of the wake, propeller loading, cavitation, appendage interactions and scale effects. In particular, quantitative information regarding the influence of local hydrodynamic flow characteristics on the acoustic performance of complete hull-propeller-appendage configurations remains limited.

Therefore, the present sudy focuses on the hydrodynamic aspects associated with underwater noise generation in a naval vessel, with particular attention to the wake field, hull pressure distribution, propeller inflow, tip and hub vortices, and the interaction between the hull, propeller and appendages. CFD simulation are used to identify the flow characteristics that may contribute to increased pressure fluctuations, cavitation and propeller-induced noise.

2.Materials and methods

2.1. Velocity distribution on the hull and wake field

The wake field is one of the most important indicator of hull-forms quality, especially for a navy vessel. In most cases, the minimizing wake-field gradiends is a primary design objective. The advantage of applying CFD methods for analyzing the wake field is the availability of the velocity components at every location in the computational damain.

The analysis of the velocity contours at successive hull stations gives an idea on the development of the boudary layer along the hull and the formation of the wake field in the stern region. The analysis of these results facilitates the identification of global or local hull-form modifications that influence the propeller inflow conditions. The calculation of the wake field should be performed for the full-scale ship.

The differences between the wake field of the model-scale and full-scale ship have a significant influence on the accuracy of the predicted acoustic behaviour of the ship (cavitation inception, unsteady propeller forces, propeller-induced pressure fluctuations). Figure 1 presents the calculated wake field for a navy vessel in model scale and full-scale. The wake of a twin screw ship is a result of the wake field caused by the appendages shaft tubes, propeller shafts, shaft brackets etc. Therefore, it is important to include all appendage in the study.(Ali & Bejan., 2010; Ali et al., 2018).

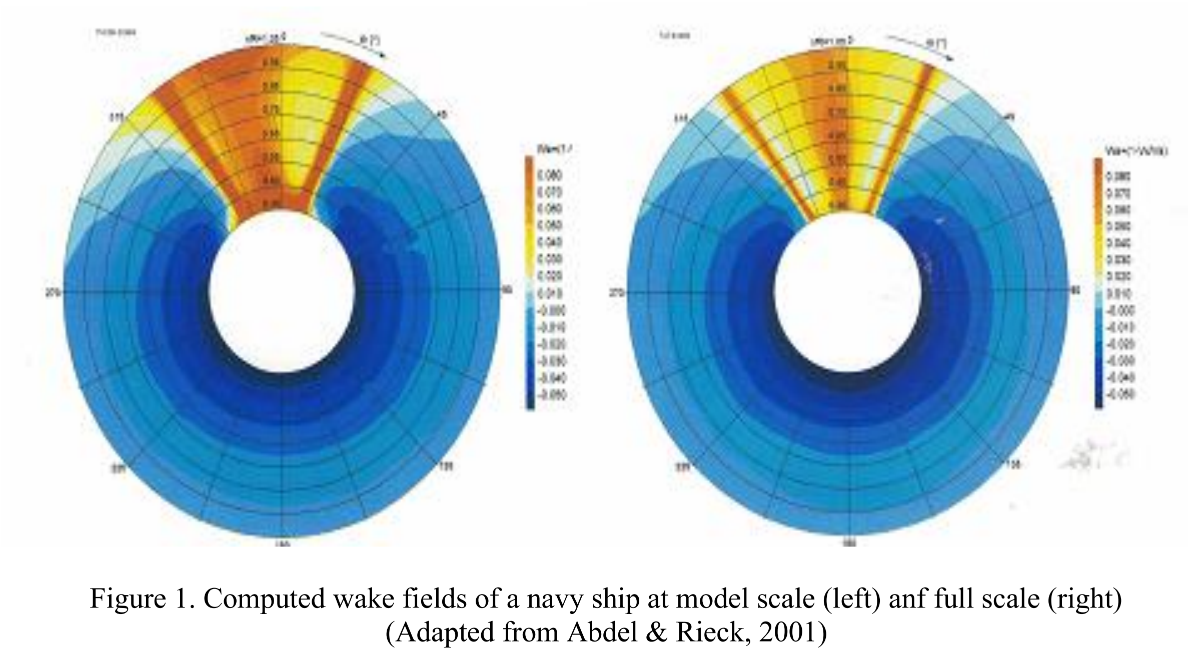

Figure 1. Computed wake fields of a navy ship at model scale (left) anf full scale (right)
(Adapted from Abdel & Rieck, 2001)

2.2. Pressure distribution on the hull and interaction between the different appendages

For a naval vessels, the problem of noise is particularly complex because the underwater signature is influenced by the complete hydrodynamic configuration of the ship. The hull form, shaft lines, shaft brackets, stabilizing fins, ruddpers, propellers and other appendages modify the propeller therefore has to be considered during the design procces rather than treating the propeller as an isolated component. The quality and uniformity of the wake field are particularly important because variations in axial and tangential velocity change the instantaneous angle of attack and loading of the propeller blades as they rotate through the wake (Smith & Rigby, 2022).

The pressure distribution on the hull gives an overview on the flow behavior around the ship. The consideration of the free surface deformation in the calculation is important for a correct analysis of the pressure field. The objective of the pressure-field analysis is to define the required hull modification in order to reduce the area of low-pressure regions on the ship hull, and also to minimize local pressure gradients. Low-pressure regions and high turbulence degrees generally have advers acoustic characteristics. It is important to extent of low-pressure regions, especially on the stern and near appendages.The flow around the appendages and the hydrodynamic interactions among appendages may be one of the significant sources of underwater noise, which should be studied with great care.

The appendages such as the struts of the propeller shaft, brackets or the stabilizing fins must be aligned with the local flow direction. The determination of the optimum orientation of the appendages in the flow several iterative calculations. Initially the flow around the ship is simulated with the propeller shaft included but without the shaft struts. Following the initial flow simulation the design of the struts can be carried out, based on information on the local direction of the flow at the location of the struts. The selected orientation is often a compromise between hydrodynamic optimum and practical design constraints. Because the struts can significantly influence on the local direction of the flow, the CFD simulation should be repeated to verify the final design and to find out if any additional modifications are required. An important advantage of CFD studies is that many configurations can be investigated in a short time.

2.3. Pressure distribution on the propeller under open water conditions

The propeller inflow has a strong influence on the acoustic behavior on the propeller. The periodic variation on the angle of attack to the propeller blades is strongly determined by the non-uniformity of the wake. The geometry of navy propellers is characterised by a moderate skew angle. The flow around a skewed propeller is more complicated than around a conventional one. Besides the usage of advanced potential flow methods based codes in the propeller design, it is important to predict the viscous flow of the propeller blades especially in the tip and in the trailing-edge regions.

The calculation of the pressure distribution within the tip-vortex core should be evaluated at full scale rahter than at model scale because of the pronounced scale effects in this region.

Figure 2 illustrates the low-pressure region in the tip vortex of a navy propeller. A steep adverse pressure gradient on the trailing edge can cause flow separation in this region. The resulting unsteady flow structures can have a serious consequence for the noise level of the propeller. Therefore, it is important to check if any flow separation takes place near the trailing edge.

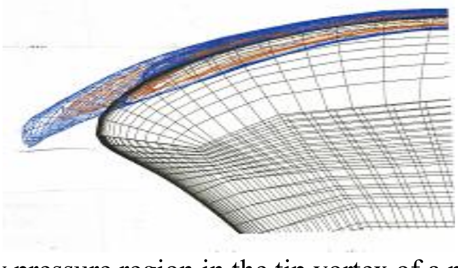

Figure 2. Low pressure region in the tip vortex of a navy propeller
(Adapted from Abdel & Rieck, 2001)

2.4. Velocity distribution in the tip and hub vortex regions

The streamline topology around the tip vortex band the trailing edge at different operating points of the propeller is an important indication for the propeller flow quality. The location at which the tip vortex detaches the propeller blade must be considered. Premature detachment of the tip vortex from the propeller blade can lead to a formation of a secondary vortex structure, which can cause a highly unsteady flow in this region.

The interaction between the blade-root vortices the form of the hub-cap geometry and its extension must be considered to intensity of the hub vortex and an delay hub-vortex cavitation inception (figure 3).

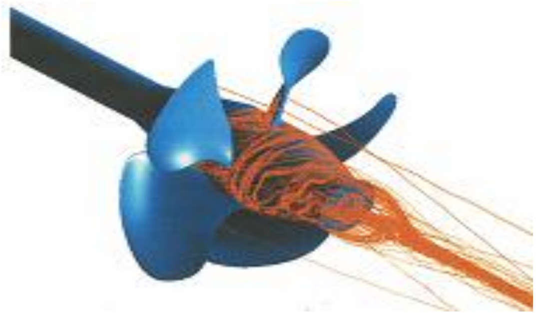

Figure 3. Formation of the hub vortex behind a navy propeller equipped with a special hub cap

3. Results and disscution

3.1. Interaction between the ship and its propeller

The primary factors, which influence the underwater noise generation by the propeller, are propeller geometry, operating conditions (propeller rotational speed, ship speed) wake-field non-uniformity, cavitation and Reynolds number as well as the water quality. The optimal adaptation of the propeller geometry to the circumferential variation of the inflow velocity is the most important criterion for reducing the propeller-induced underwater noise.

A decrease in the axial inflow velocity leads to an increase in the angle of attack of the propeller blades. Since the wake flow is directed upward in the propeller plane, the descending blade experiences a higher relative inflow velocity than the ascending blade (Figure 4).

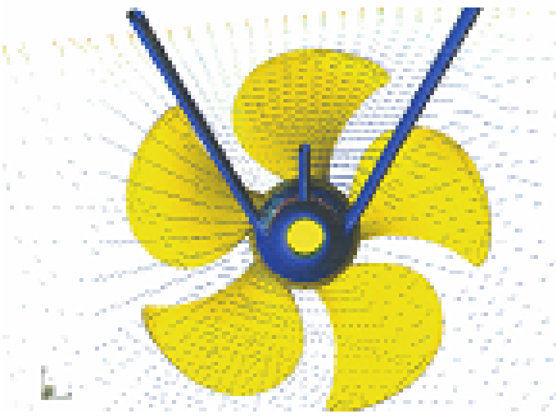

Figure 4. Radial and tangential velocity components in the propeller plane for an operating propellers

The fluctuation of the unsteady forces on the propeller and on the hull is one of the most important sources of underwater noise generated by a navy vessels. Reduction in underwater noise can be achieved by reducing the velocity gradient in the wake field, by increasing the propeller blade number and by optimising the propeller geometry (skew and blade-section shape). Extensive numerical investigation were carried out to optimise the hull form and the appendage configuration of navy vessel.

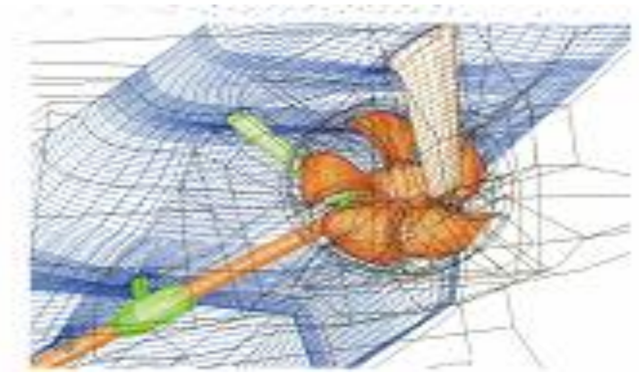

Figure 5. The computational grid for tne navy vessel with operating propellers

Figure 5 illustrates the computational grid for the navy vessel with operating propellers. The streamlines in the wake of the propellers are presented in the Figure 6.

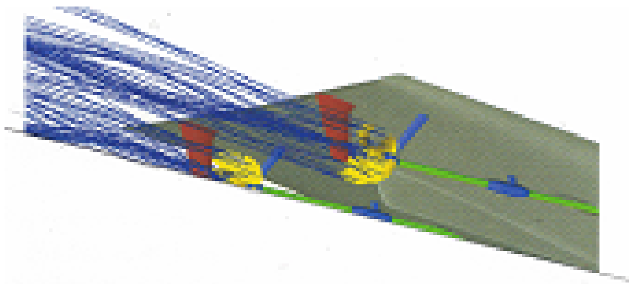

Figure 6. Streamlines in the wake of a navy ship with operating propellers

An exemple of the variation in the axial forces acting on the propeller blades is illustrated in Figure 7. The calculated propeller-induced noise levels as a function on the relative phase angle between the propellers of a twin-screw ship are shown in Figure 8.

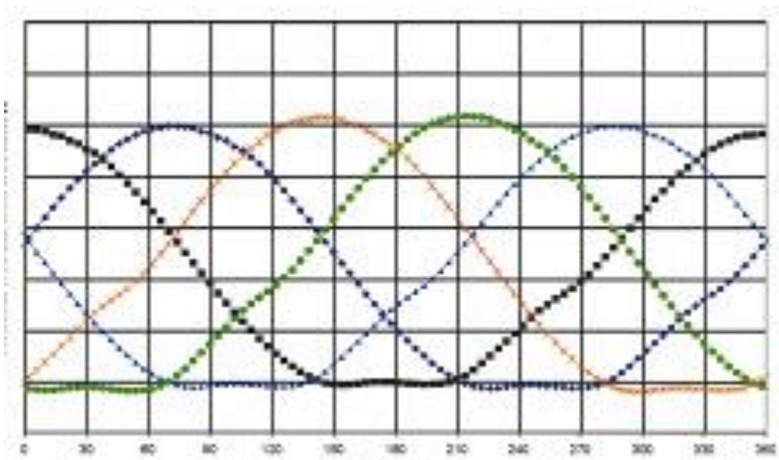

Figure 7. The variation of the axial forces acting on the propeller blades
(Adjusted and calculated in accordance with Abdel & Rieck, 2001)

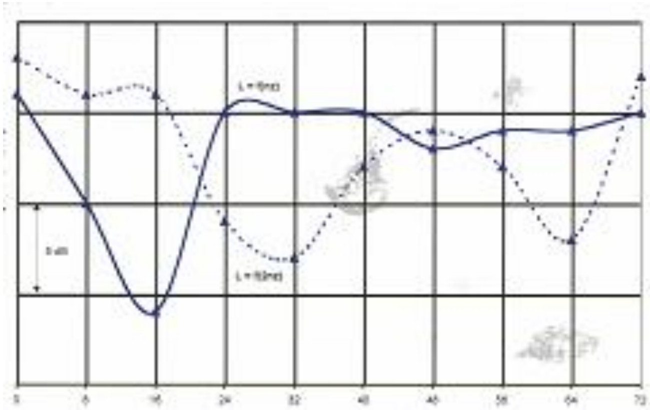

Figure 8. The variation of the calculated propeller-induced noise levels as a function of the relative phase angle between the propellers of a twin-screw ship
(Adjusted and calculated in accordance with Abdel & Rieck, 2001)

3.2. Interaction between appendages and propeller

The interaction between the propeller and the appendages should be carefully investigated. It is important to minimize disturbance of the propeller inflow resulting from the wake field of the appendages or the free vortex generated by appendages, particularly stabilizing fins and bilge keels or any other appendage.

The interaction between the propeller flow and the rudder represents a critical aspect. Analysis of the flow around the rudder is essential to avoid excessive rudder inflow angles and also to ensure that there is no very low-pressure region near the lower edge of the rudder.

4. Conclusions

The scope of the research is the optimisation of the propulsion system from the hydrodynamic point of view, with emphasis on high propulsive efficiency, reduced cavitation, reduced pressure fluctuations and lower underwater noise levels. CFD rezults are used to identify flow features that can contribute to the underwater signature and to provide guidance for the hydrodynamic optimisation of the hull, propeller and appendages.

The reduction of underwater noise is a result of fine-tuning of every detail of the local flow field around the hull, the propeller and the appendages and the interaction between each. It is very important to check every local detail of the underwater geometry of the navy vessel.

The most critical regions for generation of under–water noise are located near the openings, appendages, propellers and rudders. The hydrodynamic interactions between the hull and the propeller as well as among the various appendages should also be considered.

Author contributions:

- Conceptualization: B.A. and A.S.
- Methodology: B.A.
- Investigation: L.A.
- Writing original draft: L.A.
- Writing review and editing: B.A.
- Supervision: A.S. and L.A.

Funding: This research was not supported by any external funding.

Conflict of interest: The authors declare no conflict of interest.

References

1. Abdel Maksoud, M. & Rieck K. (2001). Calculation of the pressure reduction in the tip vortex core of a skew propeller in model and full scale. 4th Numerical Towing Tank Symposium, Hamburg, Germany, September 2001.
2 Ali, B. & Bejan. M. (2010). A new stern design - a solution of drag reduction of ships, DAAAM International Scientific Book, pag.0143-0144, ISSN 1726-9687.
3. Ali, B., Șomoiag, P., Ali, L. (2018). Establishment of criterion of water flow phenomenon around the ship, "Mircea cel Batran" Naval Academy, Scientific Bulletin, Volume XXI, Issue 1, Published by "Mircea cel Batran" Naval Academy Press, Constanta, Romania, ISSN 1454-864X, pag.531-534.
4. Smith, T. A. & Rigby, J. (2022) Underwater radiated noise from marine vessels: A review of noise reduction methods and technology Ocean Engineering, 266, 112863.
5. Sakamoto & Kamiirisa. (2018). Prediction of the near field propeller cavitation noise by viscouse CFD, with semi-empirical approach and its validation in model and full scale. Ocean Engineering 168, 41-59.
6. Kalikatzarakis, M., Coraddu, A., Altar M., Gaggero, S.,Tani, G., Villa, D., and L. Oneto, (2022). Computational prediction of underwater radiated noise of cavitating marine propellers On the accuracy of semi-empirical models, Ocean Engineering vol.259 / 111477.
7. McIntyre, D., Rahimpour, M., Dong, Z., Tani, G., Miglianti, F., Viviani, M., & Oshkai, P. (2022). Experimental measurements and numerical simulations of underwater radiated noise from a model scale propeller in uniform inflow. Ocean Engineering 255, 111409.
8. Lee, T., Ahn, B.-K., Lee, K., Lee, Y., & Kim, H.-J. (2024). Prediction on cavity inception speed and underwater radiated noise of a full scale marine propeller based on cavitation tunnel model test. Ocean Engineering, 309, 118456.