Numerical study on the scale effect of high-skew propeller E1619

L W Liu¹, M X Chen¹, Y H Wu¹*, Z G Zhang¹ and X Z Wang¹
¹Huazhong University of Science and Technology, Wuhan, China

*E-mail: m201971667@hust.edu.cn

Abstract. In this study, a large skewed propeller INSEAN E1619 model with 7 blades working in open water conditions has been numerically simulated by an inhouse URANS (Unsteady Reynolds-averaged Navier-Stokes) code coupled with dynamic overset grid approach. Grid and time-step sensitivity analysis of the propeller model has been conducted and the numerical results have been compared with available experimental data performed by INSEAN aimed to validate the numerical method. Propeller open water characteristics have been compared for models with different scales at given advance coefficient to evaluate scale effect of this high-skew propeller. Non-dimensional pressures on different scaled models have been compared between models with different scales. Results show that propeller’s thrust and torque increase with the increase of propeller scale, which is mainly due to the increase of pressure component. It could be found that the pressure on suction side near the trailing edge decreases with the increase of propeller’s scale.

1. Introduction

Propeller action is the main propulsion mode of marine vessels, and an impressively large variety of propeller designs could be found on modern ships. Accurate prediction of propeller’s performance is a perennial subject of propeller designs. Nowadays numerical methods have been widely applied on investigation of propeller’s hydrodynamic performance. Gaggero et al. [1] presented a study on the prediction of the tip and the tip leakage vortex cavitation for different propellers using a commercial RANS solver. The cavitating phenomena at the tip has been observed in experiments and numerical studies. Wang et al. [2] studied the correlation between the evolution of propeller trailing vortex wake and skew angle using four different five-blade propellers with different skew angles. Results show that the deformation of hub vortex and destabilization of the tip vortices are weakening with the increase of propeller's skew angle. Kowalczyk and Felicjancik [3] investigated the hydroacoustic characteristics of model-scale propellers under different loading conditions and the applied numerical approach was validated by presented experimental results. Paik [4] performed a numerical study on the performance of a propeller operating near a free surface. A commercial RANS solver was used for the simulations of propeller KP505 and results show that thrust of the small model propeller was underestimated compared to the experimental data while the torque was overestimated. Nouri and Mohammadi [5] numerically investigated the effect of camber ratio distribution over the propeller blades on open water performance. Results show that the location of maximum efficiency of the propeller is unchanged versus advance coefficient with the variation of camber ratio. Wang et al. [6] conducted numerical simulations on a propeller impacted by heave motions in cavitating flow using a RANS method.
Results show that the propeller load and the wake field are closely related to the variation of heave motion period.

Full-scale propeller performance is traditionally predicted from model-scale results using the standard ITTC 1978 performance prediction method [7]. More researches have indicated that this traditional method could not show the scale effect on propeller characteristics accurately, especially for the high-skew propeller [8]. The flow characteristics over the full-scale propeller are different from those at model scale, because of differences in propeller’s boundary layer generated by the increase of Reynolds number [9]. Therefore, CFD (computational fluid dynamics) method could be used in full-scale predictions and evaluating the scale effect [10]. Choi et al. [11] investigated the scale effect on the performance of a tractor type podded propeller. Numerical analysis indicates that the performance of the podded propeller blades is relatively little by Reynolds number. Bhattacharyya et al. [12] studied the scale effect on the open water characteristics of ducted propellers based on CFD simulations. Results show that the differences are related to the flow acceleration produced by the duct. Sun et al. [13] analyzed the scale effect on the evolution of propeller wake using DES (detached eddy simulation) method. Simulation results demonstrate that the full-scale propeller exhibits a stronger blade shed vortex.

Above the literature reviews, the understanding about scale effect on high skew propellers is still limited. A high skew propeller consists of more blades than a low skew propeller, aimed to reduce the cavitating noise effectively. In this study, open water propeller characteristic of a high skew propeller INSEAN E1619 was predicted using a viscous inhouse CFD approach, HUST-Ship and the simulation results were compared with experimental data. Propeller models with 5 different scales have been considered in numerical study to analyze the scale effect, focused on the differences of pressure distribution. Discussions of simulation results are presented and some conclusions are drawn finally.

2. Propeller model

A high skew propeller E1619 with 7 blades designed by INSEAN is selected as the study objective. Models with 5 different scales are considered for CFD analyses to investigate the scale effect. Geometry of the propeller is shown in figure 1 and details of the main parameters are shown in table 1.

![Figure 1. Geometry of the high skew propeller.](image)

| Table 1. Main parameters of the propeller. |
|-------------------------------------------|
| Model 1 | Model 2 | Model 3 | Model 4 | Model 5 |
| Number of blades, $N$ | $7$ | | | |
| Diameter, $D$ | 0.18m | 0.485m | 1m | 2m | 3m |
| Hub diameter ratio, $d_h/D$ | | 0.226 | | | |
| Pitch ratio at $r/R=0.7$, $P/D$ | | | 1.15 | | |

Five different scaling models are simulated in this study at open water condition and table 2 shows the simulation conditions, where $J$ is the advance coefficient defined as:

$$J = \frac{V}{nD}$$

where $V$ is the advance speed, $n$ is the rotational speed and $D$ is the diameter of propeller. The advance coefficient $J=0.85$ corresponds to the highest efficiency condition of the propeller.
As for the propeller model 2 (D=0.485m), open water experiments were performed in INSEAN towing tank [14] and the numerical results under corresponding conditions (J=0.85) could be compared with experimental data to validate the CFD method.

### Table 2. Simulation condition.

|                  | Model 1 | Model 2 | Model 3 | Model 4 | Model 5 |
|------------------|---------|---------|---------|---------|---------|
| Advance coefficient, J | 0.85    | 0.85    | 0.85    | 0.85    | 0.85    |
| Advance speed, $V_a$ (m/s) | 3.825   | 2.063   | 2.962   | 4.189   | 5.131   |
| Rotational speed, $n$ (rps) | 25      | 5       | 3.48    | 2.46    | 2.01    |
| Reynolds number, $Re$ ($10^6$) | 0.230   | 0.334   | 0.988   | 2.80    | 5.13    |

### 3. Computational method

The inhouse viscous CFD code HUST-Ship is developed for ship hydrodynamics and CFD simulations. URANS equations are solved as governing equations in liquid phase of free surface flow. Continuity and momentum conservation equations are as follows:

$$\frac{\partial \rho}{\partial t} + \frac{\partial (\rho U_i)}{\partial x_i} = 0$$  \hspace{1cm} (2)

$$\frac{\partial U_i}{\partial t} + \frac{\partial U_i U_i}{\partial x_j} = -\frac{1}{\rho} \frac{\partial \rho}{\partial x_i} + \frac{1}{\rho} \frac{\partial}{\partial x_j} \left( \mu \frac{\partial U_i}{\partial x_j} - \rho u_i u_j \right)$$  \hspace{1cm} (3)

where $U_i$ and $U_j$ ($i, j = 1, 2, 3$) represent time-averaged velocity components; $\rho$ is density; $\bar{\rho}$ is time-averaged pressure; $\mu$ is dynamic viscosity and $\rho u_i u_j$ is Reynolds stress term. This term can be calculated by using SST $k-\omega$ turbulence model to close the governing equations.

Governing equations are discretized in space using the finite difference method. Dynamic overset grid approach is applied for the connection between propeller and computational domain, allowing large-amplitude motion. PISO (pressure-implicit split-operator) algorithm is employed for the solution of pressure-velocity coupling effectively.

For the open water simulation, an appropriate computational domain is created with 7D long, 4.5D wide and 5D deep. An absolute inertial earth-fixed coordinate system is defined and the setup of boundary conditions is described in figure 2. Cartesian grid is generated in computational domain. Figure 3 shows the body-fitted structured grid divided on blades, hub and shaft of the propeller model and grids on boundary layer are small enough to yield $y' < 1$. A refinement Cartesian block closer to the propeller is used to resolve the flow field near the propeller accurately. The inhouse overset-grid program HUST-Overset is applied for assembling different grids together and generating the overset region, as shown in figure 4.

![Figure 2. Boundary conditions and grid of computational domain.](image-url)
4. Result and discussion

4.1. Verification and validation
Systematic grid and time-step convergence studies were carried out for open water simulation of propeller model 2 ($D=0.485m$) at $J=0.85$ to evaluate the convergence behavior using HUST-Ship. Three different grid schemes were used, corresponding to coarse, medium and fine grids which were refined by the constant ratio $r_G=1.414$. Three different time-steps were considered as small, medium and large time-steps with the refinement ratio $r_T=2$. Convergence analyses are based on the solutions of thrust coefficient $K_T$, torque coefficient $K_Q$, and efficiency $\eta$ of the propeller, defined as:

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

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

$$\eta = \frac{K_T J}{2 \pi K_Q}$$

where $T$ is the propeller’s thrust and $Q$ is the propeller’s torque. Figure 5 shows the results from grid and time-step convergence studies. Results of $K_T$, $K_Q$, and $\eta$ exhibit that monotonic convergence could be achieved, indicating a satisfactory convergence behavior. Simulation results are also compared with the experimental data from INSEAN, shown in figure 5, and the differences between CFD and EFD results are all less than 6%. Overall, medium grid and medium time-step could be used for the numerical study.

![Figure 5. Results from grid and time-step convergence studies.](image)

4.2. Scale effect on open water characteristics
Thrust and torque of the propellers with different scales were calculated for the comparisons. Figure 6 shows the numerical results of $K_T$, $K_Q$, and $\eta$ for different scaling propeller models. Scale effect could be observed on propeller’s thrust and torque, which increase with the increase of propeller diameter $D$. 

![Figure 6. Comparison of thrust coefficients.](image)
As propeller diameter $D<1\text{m}$, the scale effects are more obvious on thrust and torque than those when $D>1\text{m}$. On the other hand, scale effect on propeller efficiency is not significant due to the cointstantaneous increases of thrust and torque with the increase of propeller diameter.

In order to understand the reasons for the observed behaviors, friction and pressure components of propeller’s thrust and torque have been separated and shown in figure 7, where the suffixes ‘$P$’ and ‘$F$’ denote the pressure and friction components respectively. The results reveal that the contribution of friction component into thrust and torque is small but the pressure component is dominant. With the increase of propeller’s scale, the pressure component increases significantly, which mainly cause the scale effect on propeller’s thrust and torque.

Figure 8 shows the pressure distributions along blade sections ($r/R=0.7$) of different scaling propeller models, where the leading and trailing edges of propeller blade are located at 0 and 1 respectively on the position-axis. $C_p$ is pressure coefficient defined as:

$$C_p = \frac{P}{0.5\rho V^2_a}$$

(7)

It demonstrates that the pressure distribution patterns on the blades of different scaling models are similar but definite differences on suction side caused by scale effect could be observed. Pressure on suction side near the trailing edge decreases with the increase of propeller’s scale. This change leads to the increase of propeller thrust shown in figure 6.

**Figure 6.** Numerical results of $K_T$, $K_Q$, and $\eta$ for different scaling propeller models.

**Figure 7.** Friction and pressure components of propeller’s thrust and torque.

**Figure 8.** Pressure distributions along blade sections of different scaling propeller models.

5. Conclusion

In this study, the scale effect of a high skew propeller INSEAN E1619 has been analyzed using an inhouse CFD code, HUST-Ship. Systematic grid and time-step convergence studies were conducted. Simulation results have been compared with experimental data and show good agreement. Propeller models with 5 different scales were considered in numerical study, with the comparisons of open
water characteristics. Differences on pressure distribution and viscous flow field were evaluated in details. The numerical results show that propeller’s thrust and torque increase with the increase of propeller scale. As propeller diameter $D < 1\text{m}$, the scale effects are more obvious on thrust and torque than those when $D > 1\text{m}$. On the other hand, scale effect on propeller efficiency is not significant due to the cointantaneous increases of thrust and torque with the increase of propeller scale. The increase of thrust and torque is mainly due to the increase of pressure component. When analyzing the pressure distribution, it could be found that the pressure on suction side near the trailing edge decreases with the increase of propeller’s scale.

6. Future work
Future work will include CFD simulations of self-propelled submarine at both model scale and full scale using HUST-Ship.

References
[1] Gaggero S., Tani G., Viviani M., Conti F. A study on the numerical prediction of propellers cavitating tip vortex //Ocean Engineering. 2014. V. 92. 137 p.
[2] Wang L., Guo C., Su Y., Wu T. A numerical study on the correlation between the evolution of propeller trailing vortex wake and skew of propellers //International Journal of Naval Architecture and Ocean Engineering. 2018. V. 10. № 2. 212 p.
[3] Kowaleczk S., Felicjancik J. Numerical and experimental propeller noise investigations //Ocean Engineering. 2016. V. 120. 108 p.
[4] Paik K. Numerical study on the hydrodynamic characteristics of a propeller operating beneath a free surface //International Journal of Naval Architecture and Ocean Engineering. 2017. V. 9. № 6. 655 p.
[5] Mohammad Nouri N., Mohammadi S. Numerical investigation of the effects of camber ratio on the hydrodynamic performance of a marine propeller //Ocean Engineering. 2018. V. 148. 632 p.
[6] Wang L., Guo C., Su Y., Xu P., Wu T. Numerical analysis of a propeller during heave motion in cavitating flow //Applied Ocean Research. 2017. V. 66. 131 p.
[7] ITTC. ITTC Recommended Procedures and Guidelines-1978 ITTC Performance Prediction Method. In: ITTC P.C.O.T. 2017.
[8] Krasilnikov V., Sun J., Halse K.H. CFD Investigation in Scale Effect on Propellers with Different Magnitude of Skew in Turbulent Flow. First International Symposium on Marine Propulsors. Trondheim, Norway: 2009.
[9] Bhattacharyya A., Krasilnikov V., Steen S. Scale effects on open water characteristics of a controllable pitch propeller working within different duct designs //Ocean Engineering. 2016. V. 112. 226 p.
[10] Shin K.W., Andersen P. CFD Analysis of Scale Effects on Conventional and Tip-Modified Propellers. Fifth International Symposium on Marine Propulsors. Espoo,Finland: 2017.
[11] Choi J., Park H., Kim H. A numerical study of scale effects on performance of a tractor type podded propeller //International Journal of Naval Architecture and Ocean Engineering. 2014. V. 6. № 2. 380 p.
[12] Bhattacharyya A., Krasilnikov V., Steen S. A CFD-based scaling approach for ducted propellers //Ocean Engineering. 2016. V. 123. 116 p.
[13] Sun S., Wang C., Guo C., Zhang Y., Sun C., Liu P. Numerical study of scale effect on the wake dynamics of a propeller //Ocean Engineering. 2020. V. 196. 106810 p.
[14] Di Felice F., Felli M., Liefvendahl M., Swennberg U. Numerical and experimental analysis of the wake behavior of a generic submarine propeller. First International Symposium on Marine Propulsors. Trondheim, Norway: 2009.