Numerical Simulations of the Hydrodynamic Performance of the Propeller with Wake Equalizing Duct behind the Ship

Saman Rezaei¹, Milad Bamdadinejad²*, Hassan Ghassemi²

¹Department of Mechanical Engineering, Sharif University of Technology, Tehran, Iran.
²Department of Maritime Engineering, Amirkabir University of Technology, Tehran, Iran.

*Corresponding Author email: miladbamdadi@aut.ac.ir

Abstract

The equalizing wake flow into the propeller behind the ship is important from the hydrodynamic performance viewpoint. In this study, numerical simulations of the DTMB4119 propeller with two symmetric and asymmetric duct types behind the KRISO container ship (KCS) are performed using computational fluid dynamics (CFD). In order to improve the wake equaling flow, a combined duct and stators configurations were installed before the propeller in the stern of the ship and its hydrodynamic performance was studied using CFD. A duct with the NACA4415 section and two types of stator configurations are selected. The STAR-CCM+ software that uses the finite volume discretization method was used to solve the governing equations of the fluid flow. In simulating the turbulence model, the standard k-ω model was used and the solution method was validated by comparing the available experimental data. Output parameters such as thrust coefficient and torque coefficient in the open-water condition and behind the ship have been presented and discussed. Improvements in the propeller performance after mounting the asymmetric and symmetric ducts are found at 4.8% and 6.57%, respectively. So, it is concluded that the symmetric duct is more affected by the propeller performance and, hence, reduced fuel consumption considerably.

Keywords: DTMB4119 propeller; KRISO container ship (KCS); Pre-swirl stator; Symmetric and asymmetric ducts; Propeller performance; Wake equalizing duct.

Nomenclature:

\( \bar{B}_i \) Average body force
\( \bar{P}_i \) Average fluid hydrostatic pressure
\( \bar{u}_i \) Average velocity
\( \bar{u}' \) Average oscillating velocity
\( \bar{\rho}' \) Average oscillating density
\( h_i \) Basic mesh sizes
\( B_i \) Body Force
\( C_t \) Resistance coefficient
\( K_Q \) Torque coefficient
\( K_T \) Thrust coefficient
\( N_i \) The total number of cells
\( P_i \) Instantaneous hydrostatic pressure
\( P_k \) Turbulence production rate
\( Re_L \) Reynolds number based on the ship's length
\( V_M \) Model speed
\( V_S \) Ship speed
\( e_a^{ij} \) Approximated relative error
\( e_{ext}^{ij} \) Extrapolated relative error
\( r_{ij} \) Grid refinement factor
\( u_i \) Instantaneous velocity
\( u_i' \) Oscillating velocity
\( \mu_t \) Turbulent viscosity
\( \bar{\rho} \) Average density
\( \sigma_k \) Prandtl number
\( \sigma_\varepsilon \) Turbulent Schmidt number
\( \varphi_{ext}^{ij} \) Extrapolated value
\( \Delta y \) First wall thickness of the boundary layer
\( \mu \) Fluid viscosity
\( B \) Breath of KCS model
\( B \) Production buoyancy loss
\( D \) Depth of ship
\( Fn \) Froude number
\( G \) Turbulence kinetic energy
\( GCI \) Grid convergence index
\( J \) Advance coefficient
\( L \) Ship length
\( LPP \) \hspace{1em} \text{Length between perpendicualrs}
\( LWL \) \hspace{1em} \text{Length of waterline}
\( P \) \hspace{1em} \text{Hydrostatic pressure}
\( T \) \hspace{1em} \text{Draft of ship}
\( U \) \hspace{1em} \text{Velocity vector}
\( w \) \hspace{1em} \text{Wake factor}
\( k \) \hspace{1em} \text{Kinetic energy}
\( \varepsilon \) \hspace{1em} \text{Rate of dissipation of turbulent kinetic energy}
\( \eta \) \hspace{1em} \text{Efficiency}
\( \rho \) \hspace{1em} \text{Density vector}
\( \rho \dot{u}_i \dot{u}_j \) \hspace{1em} \text{Reynolds's tension}
\( \varphi \) \hspace{1em} \text{Intended parameter}

1. Introduction

Ship-owners worldwide are looking for solutions to three major challenges today: to diminish costs, to enhance efficiency and the impact on the environment. Several augmentation devices can be considered for energy-saving. It should be noted that the applicability of such devices and technologies will depend on ship type, ship size, operation profile and other factors. Thus, the decision-making for each device is required to go through the normal process of technical feasibility aspects and economic cost-effectiveness analysis for the specific ship that is under consideration.

One of the efficient devices is combined the wake equalizing duct (WED) and pre-swirl stator (PSS), as shown in Figure 1. The stator blades create a pre-swirl, giving the propeller a more favorable angle of attack. The duct increases flow velocity towards the propeller and creates a forward-directed force with its wing section shape. These features add up to possible energy savings of up to 8 percent for full-form, slower ships such as tankers and bulkers. The improvement of the propulsion efficiency of these types of ships will contribute largely to the reduction of polluting emissions and to the savings of fossil fuels in shipping [1]. According to the statistical studies on the fuel consumption of various vessels since 1980, and even 1% reduction in the fuel consumption of an oil tanker that consumes 60-100 tons/day (when moving) and imposes a daily cost of about $35-50 thousand on the owner (increasing exponentially based on the voyage length/time) is very important [2]. Figure 1. (a) shows a wake equalizing duct mounted on the ship’s stern [3] and Figure 1. (b) shows an Asymmetric duct [4]. (Figure 1)
Generally, two ways can be employed to reduce engine power. One way is to design of optimum hull which causes to decrease the resistance and the second is to improve the propulsion efficiency. To improve the thrust and efficiency, many types of energy-saving devices (ESD) are employed in the three zones of the ship stern; see ch.13 of the reference book authored by Carlton [5].

Many numerical and experimental works were carried out to analyze and design the ESD [6-11]. Using the vortex lattice method (VLM) and defining a propeller blade surface replaced by the vortex distribution. Anderson (1988) applied the VLM to obtain the hydrodynamic behavior of the propeller performance [12]. Later, many researchers worked more precisely on these methods and used the boundary element method separately to study the flow behavior in steady-state ducted propellers [13-14]. Balthazar et al. (2012) evaluated the torque and thrust of a ducted propeller in an open water mode using the panel method where the propeller hydrofoil is modeled as a well or spring. In this method, conditions considered as the problem default are closer to those governing the propeller performance in the open water and the hull and wake effects are studied on the propeller efficiency [15]. Lee et al. (2016) used a new structural safety assessment method for ESD. In this method, Morison equation was solved for a velocity at a certain probability level using two neural network methods and time domain simulation and Gamble fitting method [16]. Gaggero et al. (2018) evaluated propulsion efficiency improvements using RBCF device optimization and CFD analysis. In order to solve the RANS equations, parametric descriptions of RBCF properties and an optimization algorithm were applied in Open FOAM software. The results showed that the efficiency of the model scale has increased by about one percent, which can reach a significant amount of four percent in the unrefined propeller design [17]. Nowruzi and Najafi (2019) investigated the effects of three different pre-swirl ducts on the propulsion performance of a Series-60 ship by experimental and numerical methods [18]. Tacar et al. (2020) used CFD analysis and experiments of a new model with a larger model of a container ship to investigate the Gate Rudder system and to investigate the effect of Gate Rudder on ship performance [19]. Obwogi et al. (2021) investigated the effect of rudder bulb diameter, thrust fin, span, chord length and angle of attack on propulsion efficiency using computational fluid dynamics [20].

The boundary element method (BEM) and CFD simulation methods are widely used as important tools in the propeller design to improve efficiency performance. Ghassemi et al. published many types of research by numerical methods (BEM and CFD) to simulate the
propeller performance with different configurations under different operating conditions [21-27].

Wake equalizing duct is one of the most common energy-saving devices used to improve a ship's propulsion performance, propeller-excited vibrations, and viscous resistance forces. Many studies have been done on WED for the past three decades, most of which have been used to increase propulsion efficiency [28]. A commercial code called "Comet" was developed by Ok (2005) using the RANSE method to study the flow around a wake equalizing duct [29]. Korkut (2006) conducted a study on energy saving in powering characteristics of cargo ships using the concept of partial wake equalizing duct [30]. Celik (2007) was performed the effect of wake equalizing duct on the propulsion performance of a chemical tanker using the RANS numerical method for various WED arrangements for many ship speeds [31]. Heinke and Hellwig-Rieck (2011) investigated the effect of the Reynolds number on the flow around the appendages and on the propeller in a typical container ship by wake equalizing duct and vortex generator fins [32]. A model was developed to improve the performance of a PSS and was used to estimate power savings. In this study, the circulation distribution method was used based on the characteristics of the propellers [33]. Using a numerically variable method, Lee et al. (2019) designed an asymmetric stator using an auxiliary function by displaying the lifting lines of a vortex propeller and a stator [34]. Han (2019) estimated the energy saving efficiency of a PSS using the numerical method based on the lifting-surface method as well as the RANS equations, and the viscous flow around the hull in three systems: propeller in open water, the towing resistance experiment and the self-propulsion test [35]. In another study, the effect of angle-axis PSS was investigated using a suitable design method for each blade or location for the radius with respect to the hydrodynamic pitch angle in order to improve the propulsive efficiency in non-uniform flow fields of the stern [36]. Furcas et al. (2020) proposed a simulation-based design optimization method (SBDO) for the design of an energy-saving device based on the concept of WED [37]. Nadery and Ghassemi (2020) carried out the hydrodynamic performance of the KP505 propeller behind the KCS with and without PSS and duct. It is concluded that good design increases efficiency by 1.67%, and a bad design may reduce efficiency by 3.25% [38]. Koushan et al. (2020) numerically and experimentally investigated the effect of a pre-swirl stator (PSS) on propulsion efficiency in both model-scale and full-scale modes [39]. Recently, Nadery et al. (2021) numerically evaluated the impact of four energy-saving devices (ESDs) including pre-swirl stator (PSS), Wake equalizing duct (WED) and pre-swirl ducts called PSD-1 and PSD-2 to improve the propeller performance of the KCS container
ship [40]. In another study, they examined the effect of a new pre-swirl stator (PSS) configuration connected to the KP505 propeller on the propulsion performance of the KCS container ship [41]. Su et al. (2021) performed numerical and experimental analyzes on a 25-meter ore carrier model to investigate the operation of ESDs, including a pre-swirl stator and a rudder bulb [42]. Guo et al. (2021) used a bulk carrier scale model to study the flow mechanism of a ship and the working principles of energy-saving appendages [43]. Qin et al. (2021) numerically measured the flow effect of a pre-swirl pump jet propulsor based on improved delayed detached eddy simulation [44]. Mikkelsen et al. (2022) examined the nominal wake fields in five different heading conditions for the KCS container ship in regular waves with a wavelength equal to the length of the ship [45]. As deeply inspected in the literature, the most published works in this field are the propeller KP505 and the KCS ship hull, which the third author of the present paper worked and presented. However, this article is provided a new propeller of DTMB4119 matched with the KCS and how it works. It was our aim to find the hydrodynamic performance of this propeller by installing new duct and pre-swirl stators, which no one worked on this special propeller type.

To assess this propeller type and the proposed combination of the duct and pre-swirl stator, two kinds of symmetric and asymmetric pre-swirl stators including ducts at different velocities have been investigated. The hydrodynamic performance of the propeller (thrust and torque coefficients) under four cases (open-water, without duct, symmetric duct and asymmetric duct) at different velocities are presented and discussed.

The remainder of the paper is organized as follows. Section 2 describes the governing equations and different turbulent models. Numerical implementation, computational domains and boundary conditions are presented in Section 3. Section 4 presents the numerical results of the propeller performance under different upstream wake equalizing devices at various speeds are presented and discussed. Finally, the conclusions are drawn in Section 5.

2. Governing equations

Here, the flow simulation and propeller modeling are the main concern, and the fluid flow is studied in the control volume; therefore, a conversion tool is required that can provide its "general transfer theorem". Governing equations are used two basic hydrodynamic equations of the conservation of mass (the continuity equation) and the conservation of momentum (Navier-Stokes equations).

2.1. Continuity equation for the turbulent flow
In its general form, the continuity equation is expressed as follows:

$$\frac{\partial \rho}{\partial t} + \text{div} (\rho U) = 0$$ \hspace{1cm} (1)

In Eq. (1), \( \rho \) and \( U \) are the density and velocity vectors of the fluid, respectively. This equation is valid for the instantaneous values of the turbulent flow. The temporal averaging of Eq. (1), replacing momentary quantities with average temporal values plus the fluctuating values, and using Reynolds averaging rules, will yield Eq. (2) as follows:

$$\frac{\partial \bar{\rho}}{\partial t} + \frac{\partial}{\partial x_i} (\bar{\rho} u_i) + \frac{\partial}{\partial x_j} (\rho' u'_i) = 0$$ \hspace{1cm} (2)

In Eq. (2), \( \bar{\rho} \) and \( \rho' \) are the average density and the average oscillation density, and \( \bar{u}_i \) and \( u'_i \) are the average velocity and the average oscillating velocity, respectively. For an incompressible flow, since \( \rho' = 0 \), Eq. (2) will be as follows:

$$\frac{\partial \bar{u}_i}{\partial x_i} = 0$$ \hspace{1cm} (3)

### 2.2. Momentum equation for the turbulent flow

Momentum equations for an incompressible viscous flow are as follows:

$$\rho \left( \frac{\partial \bar{u}_i}{\partial t} + u_j \frac{\partial \bar{u}_i}{\partial x_j} \right) = B_i - \frac{\partial \bar{P}}{\partial x_i} + \mu \frac{\partial^2 \bar{u}_i}{\partial x_i \partial x_j}$$ \hspace{1cm} (4)

In Eq. (4), \( B_i \), \( P \) and \( \mu \) are the body force, fluid hydrostatic pressure and fluid viscosity, respectively. This equation is valid for both steady and turbulent flows; in the latter, dependent variables (velocity, pressure, etc.) are all time-dependent. Expressing Eq. (4) in terms of average temporal quantities, i.e., placing \( u_i = u'_i + \bar{u}_i \) and \( P_i = P'_i + \bar{P}_i \) (\( u_i \), \( u'_i \) and \( \bar{u}_i \) are called instantaneous velocity, oscillating velocity and average velocity and \( P_i \), \( P'_i \) and \( \bar{P}_i \) are called instantaneous hydrostatic pressure, oscillating hydrostatic pressure and average hydrostatic pressure, respectively) in Eq. (4) simplifying and re-applying the temporal averaging will yield Eq. (5) as follows:

$$\rho \left( \frac{\partial \bar{u}_i}{\partial t} + u_j \frac{\partial \bar{u}_i}{\partial x_j} + u'_j \frac{\partial u'_i}{\partial x_j} \right) = B_i - \frac{\partial \bar{P}_i}{\partial x_i} + \mu \frac{\partial^2 \bar{u}_i}{\partial x_i \partial x_j}$$ \hspace{1cm} (5)

In Eq. (5), \( \bar{B}_i \) and \( \bar{P}_i \) are average body force and average fluid hydrostatic pressure, respectively. Term three on the left side is usually expressed differently. Since \( \frac{\partial u'_i}{\partial x_j} = 0 \) is a
continuity equation of incompressible flows, adding and subtracting \( u_i' \frac{\partial u_i'}{\partial x_j} \) to and from the sides of Eq. (5) will result in the turbulent flow momentum equation as follows:

\[
\rho \left( \frac{\partial u_i}{\partial t} + u_j \frac{\partial u_i}{\partial x_j} \right) = \frac{\partial}{\partial x_i} \left( \mu \frac{\partial u_i}{\partial x_i} \right) + \rho \left( \frac{\partial u_i'}{\partial x_j} \right) \tag{6}
\]

The only difference between Eq. (6) and momentum equation with instantaneous quantities is in the addition of the last term, i.e., \( \rho \overline{u_i' u_j'} \), which is idiomatically called the turbulence tension or Reynolds’s tension.

2.3. Standard \( k-\varepsilon \) model

Due to its simple understanding and programming, this model is very popular. It is used with the Boussinesq relationship and expresses the turbulent field using two variables: the turbulent kinetic energy \( (k) \) (Eq. (7)) and the rate of dissipation of turbulent kinetic energy \( (\varepsilon) \) (Eq. (8)).

\[
\begin{align*}
\rho \left( \frac{\partial \overline{u_i^2}}{\partial t} + \overline{u_j u_i} \right) & = \sum_{j=1}^{n} \overline{\rho \omega_i \omega_j} + \sum_{j=1}^{n} \overline{\rho u_j u_i} \tag{7}
\end{align*}
\]

\[
\begin{align*}
\frac{\partial \varepsilon}{\partial t} & = \frac{\mu}{\rho} \left( \frac{\partial \overline{u_i' u_j'}}{\partial x_i} \right) + C_1 \frac{\varepsilon}{k} \left( \overline{u_i u_j'} \right) C_2 \frac{\varepsilon}{k} \left( \overline{u_i' u_j'} \right) - C_3 \frac{\varepsilon^2}{k} \tag{8}
\end{align*}
\]

Dimensional analyses have shown that the turbulent viscosity is directly proportional to the velocity and the scale length of large vortices in the turbulent flow field. Thus:

\[
\mu_t = C_\mu \frac{k^2}{\varepsilon} \tag{9}
\]

In Eq. (9), \( \mu_t \) is called turbulent viscosity and \( C_\mu \) is equal to 0.09. \( \varepsilon \) and \( k \) are found by semi-experimental Eqs. (10) and (11). In the following relation, \( \sigma_\varepsilon \) is the turbulent Schmidt number and equal to 1.3, \( \sigma_k \) is the Prandtl number and equal to 1, \( G \) (shear production term) is the turbulence kinetic energy produced by the average flow-turbulent flow field interaction and \( B \) is the production-buoyancy loss due to the flow fluctuating density field. \( C_1 \), \( C_2 \) and \( C_3 \) are called model constants, which are 1.44, 1.92 and -0.33 respectively.

\[
\begin{align*}
\rho \frac{\partial k}{\partial t} & + \rho u_j k_j = \left( \mu + \frac{\mu}{\sigma_k} \right) \left( \frac{\partial u_i}{\partial x_j} \right) + G + B - \rho \varepsilon \tag{10}
\end{align*}
\]

\[
\begin{align*}
\rho \frac{\partial \varepsilon}{\partial t} & + \rho u_j \varepsilon_j = \left( \mu + \frac{\mu}{\sigma_\varepsilon} \varepsilon_j \right) + C_1 \frac{\varepsilon}{k} G + C_2 \frac{\varepsilon}{k} \left( \frac{\varepsilon}{k} \right) - C_3 \mu \varepsilon \frac{\varepsilon^2}{k} \tag{11}
\end{align*}
\]

2.4. \( k-\omega \) SST model
The SST is, in fact, an optimized $k - \omega$ standard model which is itself a modified form of the Wilcox model at low Reynolds, compressibility, and shear flow dispersion effects. To use both $k - \omega$ and $k - \varepsilon$ equations, an integration function is introduced which equals 1 in areas near the wall (to activate $k - \omega$ model) and zero in those far from the wall (to activate $k - \varepsilon$ model); here, a negative point is the possibility of instability and poor convergence due to model-to-model switching.

\[
\frac{\partial (\rho k)}{\partial t} + \frac{\partial }{\partial x_j}(\rho U_j k) = \frac{\partial}{\partial x_j} \left[ (\mu + \frac{\mu_t}{\sigma_k}) \frac{\partial k}{\partial x_j} \right] + P_k - \beta \rho k \omega + P_{sb} \tag{12}
\]

\[
\frac{\partial (\rho \omega)}{\partial t} + \frac{\partial }{\partial x_j}(\rho U_j \omega) = \frac{\partial}{\partial x_j} \left[ (\mu + \frac{\mu_t}{\sigma_\omega}) \frac{\partial \omega}{\partial x_j} \right] + \frac{\alpha}{k} \rho \omega^2 P_k - \beta \rho \omega^2 + P_{\omega b} \tag{13}
\]

In Eqs. (12) and (13), $U$ is the velocity vector, $\rho$ is the density and $P_k$ is the turbulence production rate. Table 1 provides a list of the constants used in the equations. (Table 1)

3. Numerical implementation

In the CFD, the meshing of the computational domain is of great importance. It seems that more than 50% of the time is consumed on defining geometry and meshing production. CFD is known as an important tool for the design of industrial products and processes related to fluids engineering because the development of strong hardware in the 1990s led to considerable growth in this field and the numerical method entered the computation field on a larger scale.

In numerical solutions, different methods are used to solve the equations, namely the finite difference method (FDM), finite volume method (FVM), finite element method (FEM), and boundary element method (BEM). Since this study has used the finite volume method, it is described herewith its solution algorithm that involves the following four steps: a) integrating the equations governing the fluid flow on the control volume; b) discretizing and converting the integral equations into a set of algebraic equations; c) selecting the solution method of the basic equations governing the flow (Navier-Stokes and continuity equations) and d) solving the set of algebraic equations.

The STAR-CCM+ software was used to numerically simulate all parts of this study. Since the flows around the ship have high Reynolds numbers, the turbulent flow method was selected using the $k - \varepsilon$ turbulence model to ship modeling. The $y+$ values were evaluated and selected based on the turbulence model. Free surface boundaries were calculated using the volume of fluid (VOF) method. Also, the standard $k-\omega$ turbulence model and numerical method of the rotating flow of the propeller (MRF) were used for propeller modeling.
Here, the next steps to continue the work are: 1) generating the problem geometry; 2) creating the solution domain, producing a mesh or discretizing the domain; 3) explaining software settings for simulation purposes and 4) describing the general trend of the problem solution.

3.1. Test case: model of ship hull and propeller

3.1.1. DTMB4119 propeller

To calculate the propeller hydrodynamic performance from either an empirical or theoretical standpoint, it is essential to have a detailed understanding of propeller geometry and the corresponding definitions used. Depending on the type of analysis and the modeling sensitivity, it is quite necessary to design carefully and pay attention to the geometry structure of the propeller under consideration. This research has used the DTMB4119 propeller which has three blades. Due to its high accuracy and capability, the Rhino software has been used to generate the propeller geometry and make a smooth and fair surface. Figure 2 shows different views of the propeller being studied and Table 2 lists its specifications. Also, the propeller with the designed duct on the stern of the ship is shown in Figure 3. (Figure 2, Table 2 and Figure 3).

3.1.2. The KRISO Container Ship (KCS)

The KCS is a modern container ship with a bulbous bow used to explore conceptual data inflow physics as well as validate CFD models [46-47]. The KCS container ship model has been used to evaluate the propeller performance in the wake field. Figure 4 and Table 3 show the geometry and features of the model under study; the main ship has been scaled by 0.03165 coefficient. DTMB4119 and KP505 as two standard marine propellers are used for validation of numerical modeling in several types of research, although the KP505 is defined as the original propeller that used for KCS and modeling the wake pattern DTMB4119 did not use for KCS hull and we try to examine its performance with KCS in our research as innovative act. (Figure 4 and Table 3)

3.2. Computational domain

3.2.1. Computational domain of the ship model

To high problem-solving capability and accuracy as well as to simulate the flow around the ship model, the STAR-CCM+ software was selected for present calculations. Considering the studies on the modeling of the ship motion, a rectangular cube has been selected for the computational domain where the ship motion simulation is done, instead of moving the ship, based on the assumption that water flows from one side of the domain as the input and hits the ship front. This is, in fact, a simulation method of the ship motion without using highly altered meshing that speeds up the process and reduces the computational cost while
maintaining accuracy. The ship is a 2DOF object defined in the heave and pitch directions and is modeled with a two-phase model. Since the ship structure is symmetrical, only half of it has been simulated. Figure 5 shows the dimensions of the computational domain and its plan in terms of the ship length. (Figure 5)

3.2.2. **Computational domain of the propeller model**

Here, two types of domains for the propeller simulation were used i.e., cubical and cylindrical with equal distances (Figures 6 and 7), after checking the numerical results with experimental data, the cubical domain was selected due to fewer errors (see section 4). In simulations, domains are both fixed and rotating, modeling of the rotational motion of the propeller is done by the rotating disk (rotating cylinder) method and that of the water flow is based on the fixed domain that involves the rotating domain as well. (Figure 6 and Figure 7)

3.3. **Meshing of the computational domain**

Many problems encountered practically involve complex geometries for which creating a structured block mesh of hexagonal or quadrilateral cells is time-intensive. But, the computational time and, hence, the cost can be reduced using unstructured meshes with triangular-quadrilateral cells; if the geometry is simple, both ideas will take equal time. Another point worth noting in meshing and modeling is that the mesh and flow regime should match. Since convergence occurs better and faster for hexagonal-quadrilateral meshes, the highest effort has been made for all areas to be created with these meshes to save both the solution time and cost and reach more accurate solutions in the end.

Here, the emphasis is on the free water surface meshing because of the ship’s two-phase computational domain. To increase the computational accuracy in the front and rear parts of the ship, which has great effects on the output results, a separate control volume has been used to make meshes finer. The hull periphery meshes should be small enough to enable it to feel the effects of the free surface. Therefore, about 6,000,000 meshes have been used for meshing the ship hull. Specifications of the really important hull boundary layer are listed in Table 4 and meshing in the computational domain is shown in Figure 8. (Table 4 and Figure 8)

For the propeller, because it is fully submerged and no free surface is considered. Therefore, the computational domain is considered a single-phase. To increase the accuracy, finer meshes have been used in the rotating cylindrical sub-domain where it is close to the propeller blades, and to reduce the computational costs, meshes used far from the propeller are relatively larger (Figure 9).
When doing meshing and using wall functions, attention should be paid to the first node distance from the wall surface called $y^+$. If the latter is too large, the first node may lie beyond the boundary layer causing the functions to be applied too far from the wall, and if it is too small, this node will lie in the steady zone of the boundary layer where the related functions are invalid. The first wall thickness of the boundary layer ($\Delta y$) can be obtained from the approximation of Eq. (14) as follows:

$$\Delta y = \sqrt{74L}y^+ Re_L^{13/4}$$  \hspace{1cm} (14)

In Eq. (14), $L$ is the ship length and $Re_L$ is the Reynolds number based on the ship's length. However, using wall functions will reduce the separation estimation and accuracy in vortex flows. To use the SST and $k-\omega$ turbulence models, we should have $y^+ < 300$, and to benefit from low-value Reynolds models, we should have $y^+ < 2$ which is not quite easy due to the highly increased computational volume in most industrial applications. Figures 10 and 11 show the $y^+$ contour on the propeller and the ship hull, respectively. Table 5 shows the comparison of the calculated resistance coefficient ($C_t$) of the bare hull with the experimental value at the speed of 2.196 m/s ($Fn=0.26$). This speed is the service speed for the KCS which is 24 knots. It can be seen that the difference between the calculated value and the experimental value is a good agreement, in which the relative error is about 4%. (Figure 10, Figure 11 and Table 5)

### 3.4. Computational settings

Specifications of the ship, propeller, domain, solver conditions, subdomains, boundary conditions and so on are issues the determination of which requires special techniques in the software setting. Setting boundary conditions that can accurately reflect the real conditions is important to get accurate results. Boundary conditions are a set of features and conditions that need to be fully defined on the boundaries of the solution domain so that the flow is simulated.

In the solver section, the maximum iterations are determined for solving the governing equations in each time step, and the remaining is used to control the solution of the equations. The number of iterations and the acceptable remainder are important factors in the solution time. In simulating the ship motion, the solution is by the unsteady method, and for the propeller, it is by the steady method. Table 6 is given the characteristics of the applied numerical model. (Table 6)

### 3.5. Meshes of the solution strategy
The general simulation process and modeling strategy are shown in Figure 12. The simulation has had five general steps: a) modeling the ship; b) modeling the propeller in open water condition; c) modeling the propeller behind the ship; d) wake measurement before duct installation location and e) modeling the symmetrical and asymmetrical duct section behind the propeller.

(A): Modeling the ship
In this case, after entering the software and defining the ship geometry, a square box was plotted as the solution domain. Since the ship hull was symmetrical, half of it was simulated to speed up the calculations and the problem was solved by defining the center of gravity and ship velocity and setting the meshes and time steps. To get the problem outputs, a circular plate was defined and the flow velocity results were recorded on it.

(B): Modeling the propeller in open water condition
In this case, mesh settings were performed for the steady solution after defining the propeller geometry in the software and determining the solution domain (input, output, and wall), and the thrust and torque coefficients and the efficiency (according to the last recorded results) were extracted after the solution results converged.

(C): Modeling the propeller behind the ship
In this case, besides observing the solution and geometry conditions, the data recorded on the circular plate entered the simulation as the input (input flow) and, this way, the effect of the ship hull axial wake was considered in the propeller performance.

The recorded input enters in this step as an Excel file. The nature of the data is the velocity components (i, j, k) at a known local point.

(D): Wake measurement before duct installation location
In this step, the hull wake is measured again and stored before the point intended for the duct installation.

(E): Modeling the symmetrical and asymmetrical duct section behind the propeller
The duct is designed in the Rhino software based on the wake model recorded in step (B) and placed in the front part of the propeller. The problem is then solved to extract the thrust and torque coefficients. (Figure 12)

4. Results and discussion
Before presenting and analyzing the results obtained from the numerical solution of the flow on the selected ship and propeller geometries, it is necessary to check the solution accuracy and study the mesh independency related issues and, then, verify the accuracy of the results
after ensuring that they are independent of the meshing size. A very important issue in numerical solutions is to validate the obtained results by comparing them with the experimental, numerical, or semi-analytical results.

4.1. Mesh generation analysis

Mesh size and mesh number are important parameters in the numerical analyses; when meshes increase, the computational time increases too and the existing hardware fails to do the analyses. Since finer meshes increase in computational time and cost, and larger ones cause computational errors, their size and number should be so selected that the analyses may have the least error and the existing hardware can do the calculations. Meshes on the ship hull are considered much finer than on other areas because the physical phenomena affect it considerably and we also need to produce a steady and high-quality free surface. In this study, the propeller performance modeling is based on different meshing for which the thrust coefficient is a comparison factor.

In this study, three types of unstructured meshes were used to calculate the values of thrust, torque and flow field around the propeller. In this method, a trimmed mesh with 1.3 surface growth rate and fast volume growth rate has been used. Therefore, the results are obtained with a smaller number of meshes. For grid refinement, the convergence analysis method was performed for the results of mesh independence. The results were obtained from four mesh sizes (coarse, medium, fine and finer) for the propeller.

Based on the grid convergence index method (GCI) proposed by Celik et al. in 2008 [48], the mesh generation analysis was validated. In this method, the apparent order (p) and q (p) is defined as Eqs. (15) and (16) as follows:

\[ p = \frac{1}{\ln(r_{21})} \ln \left[ \frac{\varepsilon_{21}}{\varepsilon_{21}} + q(p) \right] \quad (15) \]

\[ q(p) = \ln \left( \frac{r_{21}^p - s}{r_{32}^p - s} \right) \quad (16) \]

In Eq. (16) s is calculated by \( s = l. \, \text{sign} \left( \frac{\varepsilon_{32}}{\varepsilon_{21}} \right) \).

Here, the grid refinement factor for four different meshes, i.e. (1) finer, (2) fine, (3) medium and (4) coarse, are \( r_{21} = h_2/h_1 \), \( r_{32} = h_3/h_2 \) and \( r_{43} = h_4/h_3 \) (\( h_i \) is the basic mesh size).

Moreover, for the intended parameter of \( \varphi \) (in the current study \( K_T \)), \( \varepsilon_{i+1,i} \) are calculated by \( \varepsilon_{21} = \varphi_2 - \varphi_1 \), \( \varepsilon_{32} = \varphi_3 - \varphi_2 \) and \( \varepsilon_{43} = \varphi_4 - \varphi_3 \). In this case, the extrapolated value was defined as Eq. (17):
\[
\phi_{ext}^{21} = \frac{r_2^p \phi_1 - \phi_2}{r_2^p - 1}
\]

Finally, the approximated relative error (Eq. (18)), extrapolated relative error (Eq. (19)), and fine-grid convergence index (Eq. (20)) were defined as follows:

\[
e^{-21}_a = \left| \frac{\phi_1 - \phi_2}{\phi_1} \right|
\]

\[
\phi_{ext}^{21} = \left| \frac{\phi_{ext}^{12} - \phi_1}{\phi_{ext}^{13}} \right|
\]

\[
GCI_{fine}^{21} = \frac{1.25e^{-21}_a}{r_2^p - 1}
\]

As shown in Table 7, the error rate is very small after the mesh number is 1.1e6. So, this mesh number will be enough for the propeller calculations. In Table 8, these parameters are computed for considered variables of \(K_T\). As shown in Table 8, \(N_i\) is the total number of cells and the thrust coefficient was calculated and the maximum uncertainty was obtained \(GCI_{fine}^{21} = 0.2357337073\%\). (Table 7 and Table 8)

The selected time step is calculated in such a way that, according to the International Towing Tank Conference (ITTC) recommendation [49], it can rotate between 0.5 and 2 degrees in each step. Here the time step for the propeller is 0.0005, which allows the propeller to rotate 1.9 degrees.

**4.2. Validation of the propeller results**

The open-water characteristics of the propeller of DTMB4119 are compared with experimental [50-51] data at two computational domains (cubical and cylindrical domains) as shown in Tables 9 and 10. It should be noted that the numerical model used is the same as the experimental model. In these tables, \(J\), \(K_T\), \(K_Q\) and \(\eta\) are the advance coefficient, thrust coefficient, torque coefficient and efficiency of the propeller, respectively. These parameters are defined as follows:

\[
K_T = \frac{T}{\rho n^2 D^4}, K_Q = \frac{Q}{\rho n^2 D^5}
\]

\[
\eta = \frac{J}{2\pi K_T}, J = \frac{V_A}{nD}
\]

In Eq. (21) \(n\), \(D\), \(T\) and \(Q\) are rotational speed, diameter, thrust and torque of the propeller, respectively. Also, in Eq. (22), \(V_A\) is advance velocity.
Tables 9 and 10 are compared the open-water characteristics of the DTMB4119 propeller at the cubical and cylindrical domains, respectively. The average relative errors of the thrust and torque coefficients and efficiency in the cylindrical domain are 4.1% and 8%, and 4.28%, respectively. While the average relative errors of the thrust and torque coefficients and efficiency in the cubical domain are 3.49% and 7.18%, and 3.97%, respectively. Thus, the results are indicated that the cubical domain is relatively better than the cylindrical domain. So, this domain is extended to the propeller performance in the wake field including WED (duct and stator). Figure 13 shows the comparison of the open-water characteristics of the DTMB4119 propeller in the cubical domain. At the design speed \((J=0.833)\), the results of the thrust and torque coefficients are in good agreement with the experimental value. (Table 9, Table 10 and Figure 13)

4.3. Wake field prediction
When the ship is moving the wake flow behind the ship is complicated flow and it is unsteady, non-uniform and rotational flow due to boundary layer and shape of ship hull and other factors. Each ship has a typical wake field. To predict the wake field behind the ship, the ship hull should be modeled in the computational domain. The velocity field at the propeller position can be determined. The velocities behind the ship can be predicted by using the STAR-CCM+ software. By obtaining the advance velocity behind the KCS ship, the predicted wake factor \((w)\) is calculated by \(w = 1 - V_a/V_s\). Table 11 is given the predicted values of the wake factor at different model speeds. In this table, \(Fn\), \(V_s\) and \(V_M\) are Froude number, ship speed and model speed, respectively. The predicted values of wake factors are found around 0.22~0.28 which the model speed is changed from 1.524 \((Fn=0.18)\) to 2.743 m/s \((Fn=0.32)\). All numerical results are between the ranges simulated for the wake factor values for KCS. They were in between the range \(0.25 < w < 0.28\) [52]. As an example, Figure 14 shows the axial and cross-flow velocities contour behind the ship at a model speed of 2.134 m/s \((Fn=0.25)\). The wake factor at this speed is 0.277 and advance velocity is obtained at 1.542 m/s. (Table 11 and Figure 14)

4.4. Propeller performance under the wake field
After predicting the wake field, applying it to the solution domain and checking the propeller performance, the wake field is defined in the solution domain as the input flow. Since, in the previous section, the wake field was recorded at a point before water entered the propeller, it seems, in this simulation, that the propeller is located behind the ship and its performance has been investigated in the presence of the ship hull and its resulting flow. The numerical results of propeller performance in the wake field are shown in Table 12. As given in the reference
site [53], the service speed for the KCS is 24 knots and with a scale model of 31.599, the advance velocity is 2.196 m/s. As shown in this table, the thrust and torque coefficients decrease and efficiency increase with increasing the model speed. The efficiency at the service speed (2.196 m/s for the ship model) is 0.489, which seems to be low efficiency with this type of DTMB4119 propeller. (Table 12)

4.5. Symmetric and asymmetric ducts

The need to reduce fuel consumption and improve the environmental conditions led to the design of the desired duct and stator to the ship, adapted from similar installed cases in the industry. In the present paper, a duct with NACA4415 section and two types of stators (symmetrical and asymmetrical) is selected. Figure 15 shows an overview of the duct and symmetrical duct mounted at the stern of the ship. Figure 16 shows the two types of the duct and stator, called symmetrically and asymmetrically and reveals sit effects on the propeller performance. The duct has NACA4415 section type and the stator is NACA0012 section type. The symmetric type has 5 stators placed inside the duct by symmetric relative to the vertical axis, while the asymmetric type has 4 stators arranged 3 stators on the right side and one stator on the left side. (Figure 15 and Figure 16)

4.6. Hydrodynamic effect

As it is evident that the duct causes to equalize the flow into the propeller, so improve the axial velocity. Here, the thrust and torque are presented in Table 13 at four cases (open-water, without duct, symmetrical duct and asymmetrical duct) and three velocities. Since the advance velocity in open-water is bigger than wake flow, the thrust and torque coefficients are larger than the behind the ship. On the other hand, three cases behind the ship (without duct, symmetric duct and asymmetric duct) show that the thrust and torque for the symmetric duct are obtained bigger values relative to two other cases (asymmetric duct and without duct). In order to make better displaying, these data are shown in Figures 17 and 18. As inspected in the results, the symmetric duct is gained by 6.57%, while the gain of the asymmetric duct is 4.8%. (Table 13, Figure 17 and Figure 18)

5. Conclusions

This research studied the WED and stator mounted in front of the DTMB4119 propeller behind the KCS. The purpose of this paper was to find the hydrodynamic performance of the DTMB4119 propeller by installing new ducts and stators that no one worked on this propeller combined with the stator and duct. To validate the present numerical results, the available experimental data were used for the propeller under the open-water condition and resistance
coefficient of the KCS. Two computational domains of the cubical and cylindrical are used. Thrust and torque coefficients of the propeller were determined at different advance coefficients. Then, the propeller is investigated under the wake flow of the KCS. After predicting the wake behind the ship, two types of ducts (symmetrical and asymmetrical) were mounted in front of the propeller. The following conclusions are drawn from the finding of this study:

- At the model speed of 2.196 m/s (as a service speed of the KCS), the resistance coefficient is shown in good agreement between the present numerical result and experimental value.
- For propeller open-water calculations, two computational domains of the cubical and cylindrical are used. It is concluded that the cubical domain causes less error in the numerical solution than the cylindrical domain. The average relative errors for the thrust and torque coefficients and efficiency are 3.49% and 7.18%, and 3.97% in the cubical domain, respectively.
- The efficiency is reduced when the propeller is operating behind the ship compared to the open-water condition. At the service speed of 2.196 m/s, the efficiency is 0.497 and 0.489 at the open-water condition and behind the ship, respectively.
- Improvements in the propeller performance after mounting the asymmetric and symmetric ducts are found 4.8% and 6.57%, respectively.

References

[1]. Ziarati, R., Bhuiyan, Z., De Melo, G., Koivisto, H., Lahiry, H., OzTurker, E. and Akdemir, B. “MariEMS Train the Trainee (MariTTT) Courses on Energy Efficient Ship Operation”, Technical Report, (2017).

[2]. Choi, Y.B. “2008.11 DSME Ship Energy Saving Devices: Hydrodynamics R&D Team”, In The House of Wisdom and Innovation, Technical report, (2008).

[3]. Skipsrevyen Company. The green solution below the waterline, https://www.skipsrevyen.no/article/the-green-solution-below-the-waterline (2010, accessed 15 February 2018).

[4]. Wärtsilä Encyclopedia of Marine Technology. Power-Saving Devices,
https://www.wartsila.com/encyclopedia/term/power-saving-devices (2017, accessed 14 August 2019)

[5]. Carlton, J. “Marine Propellers and Propulsion”, 3th Edn., Butterworth Heinemann, J.S., (2012).

[6]. Park, S., Oh, G., Rhee, S. H., Koo, B. Y., and Lee, H. “Full scale wake prediction of an energy saving device by using computational fluid dynamics”, Ocean Engineering, 101, pp. 254-263 (2015).

[7]. Kim, J. H., Choi, J. E., Choi, B. J., Chung, S. H., and Seo, H. W. “Development of energy-saving devices for a full slow-speed ship through improving propulsion performance”, International Journal of Naval Architecture and Ocean Engineering, 7(2), pp. 390-398 (2015).

[8]. Mizzi, K., Demirel, Y. K., Banks, C., Turan, O., Kaklis, P., and Atlar, M. “Design optimisation of Propeller Boss Cap Fins for enhanced propeller performance”, Applied Ocean Research, 62, pp. 210-222 (2017).

[9]. Go, J. S., Yoon, H. S., and Jung, J. H. “Effects of a duct before a propeller on propulsion performance”, Ocean Engineering, 136, pp. 54-66 (2017).

[10]. Gaggero, S., Villa, D., Tani, G., Viviani, M., and Bertetta, D. “Design of ducted propeller nozzles through a RANSE-based optimization approach”, Ocean Engineering, 145, pp. 444-463 (2017).

[11]. Chang, X., Sun, S., Zhi, Y., and Yuan, Y. “Investigation of the effects of a fan-shaped Mewis duct before a propeller on propulsion performance”, Journal of Marine Science and Technology, 24(1), pp. 46-59 (2019).

[12]. Andersen, S. V. "Numerical treatment of the design-analysis problem of ship propellers using vortex lattice methods", PhD thesis, DTU, Lyngby, Denmark (1988).

[13]. Baltazar, João, and J. A. C. Falcão de Campos. "On the modelling of the flow in ducted propellers with a panel method", In Proceedings of the 1st International Symposium on Marine Propulsors, 9, pp. 1-15 (2009).

[14]. Cai, H. P., Su, Y. M., Li, X., and Shen, H. L. “Using the surface panel method to predict the steady performance of ducted propellers”, Journal of marine science and application, 8(4), pp. 275-280 (2009).

[15]. Baltazar, J., Falcão de Campos, J. A. C., and Bosschers, J. “Open-water thrust and torque predictions of a ducted propeller system with a panel method”, International Journal of
Rotating Machinery, 2012 (2012).

[16]. Lee, D., Jang, B. S., and Kim, H. “Development of procedure for structural safety assessment of energy saving device subjected to nonlinear hydrodynamic load”, Ocean Engineering, 116, pp. 165-183 (2016).

[17]. Gaggero, S. “Design of PBCF energy saving devices using optimization strategies: A step towards a complete viscous design approach”, Ocean Engineering, 159, pp. 517-538 (2018).

[18]. Nowruzi, H. and Najafi, A. “An experimental and CFD study on the effects of different pre-swirl ducts on propulsion performance of series 60 ship”, Ocean Engineering, 173, pp. 491-509 (2019).

[19]. Tacar, Z., Sasaki, N., Atlar, M and Korkut, E. “An investigation into effects of Gate Rudder® system on ship performance as a novel energy-saving and maneuvering device”, Ocean Engineering, 218, p. 108250 (2020).

[20]. Obwogi, E. O., Shen, H. L. and Su, Y. M. “The design and energy saving effect prediction of rudder-bulb-fin device based on CFD and model test”, Applied Ocean Research, 114, p. 102814 (2021).

[21]. Ghasemi, H. “The effect of wake flow and skew angle on the ship propeller performance”, Scientia Iranica, 16(2), pp. 149-158 (2009).

[22]. Ghassemi, H., Mardan, A. and Ardeshir, A. “Numerical analysis of hub effect on hydrodynamic performance of propellers with inclusion of PBCF to equalize the induced velocity”, Polish Maritime Research, 19(2 (73)), pp. 17-24 (2012).

[23]. Shamsi, R. and Ghassemi, H. “Time-accurate analysis of the viscous flow around puller podded drive using sliding mesh method”, Journal of Fluids Engineering, 137(1) (2015).

[24]. Majdfar, S., Ghassemi, H., Forouzan, H. and Ashrafi, A. “Hydrodynamic prediction of the ducted propeller by CFD solver”, Journal of Marine Science and Technology, 25(3) (2017).

[25]. Esmailian, E., Ghassemi, H. and Zakerdoost, H. “Systematic probabilistic design methodology for simultaneously optimizing the ship hull-propeller system”, International Journal of Naval Architecture and Ocean Engineering, 9(3), pp.
[26]. Mahmoodi, K., Ghassemi, H. and Nowruzii, H. “Obtaining mathematical functions of the propeller thrust and torque coefficients fluctuations at non-uniform wake flow including geometry effects”, Mechanics & Industry, 19(2), 205 (2018).

[27]. Zakerdoost, H., and Ghassemi, H. “Hydrodynamic optimization of ship’s hull-propeller system under multiple operating conditions using MOEA/D”, Journal of Marine Science and Technology, 26(2), pp. 419-431 (2020).

[28]. Martinas, G. “Cavitation of a Propeller and Influence of a Wake Equalizing Duct”, TransNav: International Journal on Marine Navigation and Safety of Sea Transportation, 9(2) (2015).

[29]. Ok, J. P. and TU Hamburg-Harburg. “Power savings by wake equalizing ducts?”, Ship Technology Research, 52(1), pp. 34-51 (2005).

[30]. Korkut, E. “A case study for the effect of a flow improvement device (a partial wake equalizing duct) on ship powering characteristics”, Ocean Engineering, 33(2), pp. 205-218 (2006).

[31]. Çelik, F. “A numerical study for effectiveness of a wake equalizing duct”, Ocean Engineering, 34(16), pp. 2138-2145 (2007).

[32]. Heinke, Hans-Jürgen and Katrin Hellwig-Rieck. “Investigation of Scale Effects on Ships with a Wake Equilizing Duct or with Vortex Generator Fins”, In Second International Symposium on Marine Propulsors, Hamburg, Germany, (2011).

[33]. Streckwall, H. and Xing-Kaeding, Y. “On the working principle of pre-swirl stators and on their application benefit and design targets”, International Shipbuilding Progress, 63(3-4), pp. 87-107 (2017).

[34]. Lee, K. J., Lee, H. D. and Choi, S. H. “A design and validation study of a non-axisymmetric preswirl stator”, Ocean Engineering, 189, p. 106365 (2019).

[35]. Han, C. “Optimization and Design Research on Marine Pre-swirl Stator-propeller by CFD”, Master thesis, Harbin Engineering University, Harbin, China (2019).

[36]. Lee, K. S., Kim, M. C., Shin, Y. J. and Kang, J. G. “Design of asymmetric pre-swirl stator for KVLCC2 considering angle of attack in non-uniform flow fields of the stern”, Journal of the Society of Naval Architects of Korea, 56(4), pp. 352-360 (2019).

[37]. Furcas, F., Vernengo, G., Villa, D. and Gaggero, S. “Design of wake equalizing ducts
using RANSE-based SBDO”, *Applied Ocean Research*, 97, p. 102087 (2020).

[38]. Nadery, A., and Ghassemi, H. “Numerical investigation of the hydrodynamic performance of the propeller behind the ship with and without WED”, *Polish Maritime Research*, (2020).

[39]. Koushan, K., Krasilnikov, V., Nataletti, M., Sileo, L. and Spence, S. “Experimental and numerical study of pre-swirl stators PSS”, *Journal of Marine science and Engineering*, 8(1), 47 (2020).

[40]. Nadery, A., Ghassemi, H. and Nowruzi, H. “Enhancement of the ship propeller hydrodynamic performance by different energy-saving devices mounted at the upstream zone”, *Journal of the Brazilian Society of Mechanical Sciences and Engineering*, 43(10), pp. 1-25 (2021).

[41]. Nadery, A., Ghassemi, H. and Chybowski, L. “The effect of the PSS configuration on the hydrodynamic performance of the KP505 propeller behind the KCS”, *Ocean Engineering*, 234, p. 109310 (2021).

[42]. Su, Y. M., Lin, J. F., Zhao, D. G., Guo, C. Y. and Guo, H. “Influence of a pre-swirl stator and rudder bulb system on the propulsion performance of a large-scale ship model”, *Ocean Engineering*, 218, p. 108189 (2020).

[43]. Guo, C. Y., Xu, P., Han, Y., Zhou, G. L., Kuai, Y. F. and Fan, Y. W. “Working mechanism of pre-swirl stator based on stereoscopic particle image velocimetry”, *Ocean Engineering*, 236, p. 109442 (2021).

[44]. Qin, D., Huang, Q., Pan, G., Shi, Y., Han, P. and Dong, X. “Effect of the duct and the pre-swirl stator on the wake dynamics of a pre-swirl pumpjet propulsor”, *Ocean Engineering*, 237, p. 109620 (2021).

[45]. Mikkelsen, H., Shao, Y. and Walther, J. H. “Numerical study of nominal wake fields of a container ship in oblique regular waves”, *Applied Ocean Research*, p. 102968 (2022).

[46]. Carrica, P. M., Fu, H. and Stern, F. “Computations of self-propulsion free to sink and trim and of motions in head waves of the KRISO Container Ship (KCS) model”, *Applied ocean research*, 33(4), pp. 309-320 (2011).
[47]. Feng, D., Ye, B., Zhang, Z. and Wang, X. “Numerical Simulation of the Ship Resistance of KCS in Different Water Depths for Model-Scale and Full-Scale”, Journal of Marine Science and Engineering, 8(10), 745 (2020).

[48]. Celik, I. B., Ghia, U., Roache, P. J. and Freitas, C. J. “Procedure for estimation and reporting of uncertainty due to discretization in CFD applications”, Journal of fluids Engineering-Transactions of the ASME, 130(7) (2008).

[49]. ITTC Proceedings, “Practical Guidelines for Ship CFD Applications ITTC-Recommended Procedures and Guidelines”, Section 7.5-03-02-03, International Towing Tank Conference, (2014).

[50]. Sezen, S., Doğrul, A. and Şakir, B. A. L. “An empirical approach for propeller tip vortex cavitation noise”, Sigma Journal of Engineering and Natural Sciences, 36(4), pp. 1127-1139 (2018).

[51]. Boumediene, K. and Belhenniche, S. E. “Numerical analysis of the turbulent flow around DTMB 4119 marine propeller”, Int. J. Mech. Aerosp. Ind. Mechatron. Manuf. Eng, 10, pp. 347-351 (2016).

[52]. Kinaci, O. K., Gokce, M. K., Alkan, A. D. and Kukner, A. “On self-propulsion assessment of marine vehicles”, Brodogradnja: Teorija i praksa brodogradnje i pomorske tehnike, 69(4), pp. 29-51 (2018).

[53]. Workshop on verification and validation of ship maneuvering simulation methods (SIMMAN 2008), Copenhagen, Denmark, 14-16 April 2008, MOERI Container Ship (KCS), http://www.simman2008.dk/KCS/kcs_geometry.htm (2012, accessed 10 March 2019).

List of Captions

Figure 1. (a) Wake equalizing duct mounted on the ship’s stern, (b) Asymmetric duct.

Figure 2. Different view of DTMB4119 propeller.

Figure 3. Propeller with the duct in the stern.

Figure 4. Different view of the KCS model.

Figure 5. Schematic of the computational domain and boundary conditions of the ship.

Figure 6. Rectangular cubic computational domain and boundary conditions of the propeller with the rotating domain.
Figure 7. Cylindrical computational domain and boundary conditions of the propeller with the rotating domain.
Figure 8. The meshing of the computational domain.
Figure 9. Computerization of propeller's computational domain.
Figure 10. The $y^+$ contour on the propeller.
Figure 11. The $y^+$ contour on the ship hull.
Figure 12. The general simulation process and modeling strategy.
Figure 13. Comparison of the open-water characteristics of the DTMB4119 propeller (cubical domain).
Figure 14. Axial and cross-flow velocities contours behind the ship at the model speed of 2.134 m/s, ($Fn=0.25$): (a) axial velocity, (b) along the y-axis and (c) along the z-axis.
Figure 15. Overview of the symmetrical duct.
Figure 16. a) Symmetric duct b) Asymmetric duct.
Figure 17. Comparison of the thrust coefficient under different conditions (open-water, without duct, symmetrical duct and asymmetrical duct) at three advance velocities.
Figure 18. Comparison of the torque coefficient under different conditions (open-water, without duct, symmetrical duct and asymmetrical duct) at three advance velocities.
Table 1. The stability used in the turbulence model.
Table 2. Principal parameters of DTMB4119 propeller.
Table 3. Principal parameters of the KCS model.
Table 4. Specifications of the boundary layer.
Table 5. Comparison of resistance coefficient at model speed $V_M=2.196$ m/s ($Fn=0.26$).
Table 6. Numerical characteristics used in the software.
Table 7. Mesh independency and mesh size effect.
Table 8. Discretization error for propeller thrust (T) based on grid convergence method.
Table 9. Comparison of the open-water characteristics of the DTMB4119 propeller (cubical domain).
Table 10. Comparison of the open-water characteristics of the DTMB4119 propeller (cylindrical domain).
Table 11. Predicted values of wake filed behind the KCS.
Table 12. Propeller performance in the wake field.
Table 13. Propeller performance at four cases (open-water, without duct, symmetrical duct and asymmetrical duct).
Figures

Figure 1
Figure 12

Figure 13
Figure 14
Figure 15

(a)

Figure 16

(b)
### Tables

#### Table 1

| $\alpha$ | $\beta$ | $\beta'$ | $\sigma_k$ | $\sigma_{o_i}$ |
|----------|---------|----------|------------|---------------|
| 5.9      | 0.075   | 0.09     | 2          | 2             |

#### Table 2

|                        |         |
|------------------------|---------|
| Propeller diameter (m) | 0.3048  |
| Expanded area ratio (EAR) | 0.607   |
| Number of blades       | 3       |
| Pitch ratio            | Variable|
| Rake (deg)             | 0       |
| Skew (deg)             | 0       |
| Blade section          | NACA66 a=0.8 |
| Rotation direction     | Right   |

#### Table 3

| Hull parameter                      | Value  |
|-------------------------------------|--------|
| Length between perpendicular, LPP (m) | 7.2786 |
| Length of water line, LWL (m)       | 7.357  |
| Breadth, B (m)                      | 1.019  |
| Depth, D (m)                        | 0.6013 |
| Draft, T (m)                        | 0.3418 |
| Wetted area surface (m$^2$)          | 9.4379 |
| Midship section coef. [-]            | 0.9849 |
| Block coef. [-]                     | 0.6505 |

#### Table 4

|                        |         |
|------------------------|---------|
| Number of layers       | 12 layers |
| First layer thickness  | 0.1 mm   |
| The overall thickness of the boundary layer | 3 mm |

#### Table 5

| Resistance Coef. | Calculated | Exp. | Error (%) |
|------------------|------------|------|-----------|
| $10^3C_i$        | 3.415      | 3.554| 4.1%      |
### Table 6

| Characteristics       | Propeller modeling | Ship modeling |
|-----------------------|--------------------|---------------|
| The Solver            | Open water         | Behind the ship |
| Time Step             | 0.0005             | 0.04          |
| Number of loops in    | -                  | 10            |
| each time step        |                    |               |
| Turbulent model       | \( k - \omega \) \( SST \) | \( k - \varepsilon \) |

### Table 7

| Number of mesh (N)   | \( K_T \)          | Basic mesh size (m) | Mesh size increase coefficient |
|----------------------|--------------------|---------------------|-------------------------------|
| 750,000              | 0.153548792        | -                   | -                             |
| 800,000              | 0.152624435        | 0.65%               | 0.016                         | 1.062                        |
| 1,100,000            | 0.151853246        | 0.65%               | 0.012                         | 1.27                         |
| 1,410,000            | 0.151264583        | 0.38%               | 0.011                         | 1.21                         |

### Table 8

| \( N \) (finer)     | \( N \) (medium)  | \( N \) (coarse)   | \( q \)            | \( e_{ext}^{21} \) |
|---------------------|-------------------|--------------------|---------------------|---------------------|
| \( N_1 \) (finer)   | 1,410,000         | \( N_3 \) (medium) | 800,000             | -1.389653346       |
| \( N_2 \) (fine)    | 1,100,000         | \( N_4 \) (coarse) | 750,000             | -0.151264583       |
| \( h_1 \)           | 0.011             | \( h_3 \)          | 0.016               | 0.151746354        |
| \( h_2 \)           | 0.012             | \( h_4 \)          | 0.022               | 0.151746354        |
| \( r_{21} \)        | 1.09              | \( r_{32} \)       | 1.33                | 0.151746354        |
| \( r_{43} \)        | 1.375             | \( \varphi_1 \)    | 0.151264583         | 0.389161156%       |
| \( \varphi_2 \)     | 0.151853246        | \( \varphi_3 \)    | 0.152624435         | 0.389161156%       |
| \( \varphi_4 \)     | 0.153548792        | \( \varepsilon_{21} \) | 0.000588663         | 0.389161156%       |
| \( \varepsilon_{32} \) | 0.000771189      | \( \varepsilon_{43} \) | 0.00924357          | 0.389161156%       |
| \( q \)             | -1.389653346       | \( p \)            | 12.991455817        | 0.389161156%       |
| \( e_{ext}^{21} \)  | 0.236572241%       | \( GCI_{fine}^{21} \) | 0.2357337073%       | 0.389161156%       |

35
### Table 9

| $J$ | $K_T\ (\text{Num})$ | $K_T\ (\text{Exp})$ | $10K_Q\ (\text{Num})$ | $10K_Q\ (\text{Exp})$ | $\eta\ (\text{Num})$ | $\eta\ (\text{Exp})$ |
|-----|---------------------|---------------------|-----------------------|-----------------------|----------------------|----------------------|
| 0.5 | 0.301               | 0.281               | 0.495                 | 0.463                 | 0.484                | 0.483                |
| 0.7 | 0.218               | 0.207               | 0.389                 | 0.363                 | 0.624                | 0.635                |
| 0.833 | 0.16                | 0.155               | 0.301                 | 0.28                  | 0.704                | 0.734                |
| 0.9 | 0.125               | 0.123               | 0.263                 | 0.243                 | 0.681                | 0.725                |
| 1.1 | 0.037               | 0.037               | 0.122                 | 0.112                 | 0.537                | 0.578                |
| Ave. error% | 3.49% | 7.18% | 3.97% |

### Table 10

| $J$ | $K_T\ (\text{Num})$ | $K_T\ (\text{Exp})$ | $10K_Q\ (\text{Num})$ | $10K_Q\ (\text{Exp})$ | $\eta\ (\text{Num})$ | $\eta\ (\text{Exp})$ |
|-----|---------------------|---------------------|-----------------------|-----------------------|----------------------|----------------------|
| 0.5 | 0.308               | 0.281               | 0.515                 | 0.463                 | 0.483                | 0.483                |
| 0.7 | 0.214               | 0.207               | 0.389                 | 0.363                 | 0.635                | 0.635                |
| 0.833 | 0.159               | 0.155               | 0.305                 | 0.28                  | 0.734                | 0.734                |
| 0.9 | 0.124               | 0.123               | 0.263                 | 0.243                 | 0.725                | 0.725                |
| 1.1 | 0.039               | 0.037               | 0.121                 | 0.112                 | 0.578                | 0.578                |
| Ave. error% | 4.1% | 8% | 4.28% |

### Table 11

| $V_s\ (kts)$ | $V_m\ (m/s)$ | $Fn$ | $w$ |
|--------------|--------------|------|-----|
| 16.83        | 1.524        | 0.18 | 0.278 |
| 21.59        | 1.955        | 0.23 | 0.278 |
| 23.56        | 2.134        | 0.25 | 0.277 |
| 24.00        | 2.196        | 0.26 | 0.275 |
| 28.03        | 2.539        | 0.30 | 0.229 |
| 30.29        | 2.743        | 0.32 | 0.232 |
Table 12

| $V_M$ (m/s) | $w$ | $V_A$ (m/s) | $J$ | $K_T$ | $10K_Q$ | $\eta$ |
|-------------|-----|-------------|-----|-------|---------|--------|
| 1.524       | 0.278 | 1.1         | 0.36 | 0.351 | 0.566   | 0.355 |
| 2.134       | 0.277 | 1.542       | 0.506| 0.277 | 0.465   | 0.477 |
| 2.196       | 0.275 | 1.592       | 0.522| 0.266 | 0.453   | 0.489 |
| 2.539       | 0.235 | 1.940       | 0.63 | 0.212 | 0.380   | 0.560 |
| 2.743       | 0.232 | 2.105       | 0.69 | 0.186 | 0.345   | 0.593 |

Table 13

| Behind the ship | Open-water |
|-----------------|------------|
| Asymmetric duct | Symmetric duct | Without duct | Open-water |
| $10K_Q$ | $K_T$ | $10K_Q$ | $K_T$ | $10K_Q$ | $K_T$ | $V_A$ (m/s) | $w$ | $V_M$ (m/s) |
| 0.477 | 0.285 | 0.487 | 0.292 | 0.465 | 0.275 | 0.511 | 0.306 | 1.542 | 0.277 | 2.134 |
| 0.465 | 0.277 | 0.473 | 0.283 | 0.453 | 0.266 | 0.499 | 0.298 | 1.592 | 0.275 | 2.196 |
| 0.392 | 0.222 | 0.395 | 0.226 | 0.380 | 0.212 | 0.428 | 0.244 | 1.940 | 0.229 | 2.539 |

**ORCID iD**

Saman Rezaei, https://orcid.org/0000-0002-2360-7679.
Milad Bamdadinejad, https://orcid.org/0000-0003-4106-8880.
Hassan Ghassemi, https://orcid.org/0000-0002-6201-346X.

**Biography**

**Saman Rezaei** received his master degree in Naval Architecture from Amirkabir University of Technology (Tehran Polytechnic). He is currently a Ph.D. student of maritime engineering at department of mechanical engineering in Sharif University of Technology. He has research on Marine Energy Device (ESD) and his current research interests include Marine Autonomous Surface Ship (MASS) Collision Avoidance methods base on COLREG.
**Milad Bamdadinejad** received the B.Sc. degree in Civil-Surveying Engineering from department of geodesy and geomatics engineering at Tafresh University in 2014 and the M.Sc. degree in Coastal Engineering from department of maritime engineering at Amirkabir University of Technology (Tehran Polytechnic) in 2018. His research interests include application of remote sensing and GIS in coastal engineering, marine renewable energy and coastal sediment transport modeling.

**Hassan Ghassemi** is a professor at Amirkabir University of Technology (AUT). His research interests include ocean renewable energy, marine propulsor design, dynamics of offshore structures, high-speed craft, and hydrodynamic numerical methods such as BEM and CFD. His website is: [http://hmaa.itgo.com/ghassemi.html](http://hmaa.itgo.com/ghassemi.html)