Computers & Fluids 51 (2011) 35–47

Contents lists available at SciVerse ScienceDirect

Computers & Fluids

journal homepage: www.elsevier.com/locate/compfluid

# Full scale self-propulsion computations using discretized propeller for the KRISO container ship KCS

Alejandro M. Castro, Pablo M. Carrica *, Frederick Stern

IIHR – Hysdroscience and Engineering, The University of Iowa, Iowa City, IA 52242, USA

ARTICLE INFO

*Article history:*
Received 11 November 2010
Received in revised form 11 July 2011
Accepted 12 July 2011
Available online 5 August 2011

*Keywords:*
Self-propulsion
Full scale ship
CFD
Propellers
Overset grids
Free surface flows

ABSTRACT

Self-propulsion computations of the KCS containership are performed in full-scale with direct discretization of the propeller. A dynamic overset approach is used, which allows for arbitrary rotational speed of the propeller during the computation. The self-propulsion point is obtained using a controller to modify the propeller RPS until the target speed is reached. To obtain propulsion coefficients the open-water curves of the propeller and a towed, unpropelled case are also computed. Together, these computations provide for a complete CFD prediction of self-propulsion factors at full scale. The main differences with a similar model scale simulation following the ITTC procedures are identified and reported. The effect of these differences in the propeller operation point and performance are thoroughly studied and discussed. It is concluded that for this case the propeller operates more efficiently in full scale and is subject to smaller load fluctuations.

## 1. Introduction

CFD has been used in attempts to predict self-propulsion and powering characteristics with good success. Due to the small time steps and high-cost required in a discretized propeller approach, most self-propulsion simulations are performed using a body force model of the propeller. The level of complexity of these body force approaches varies from a prescribed body force [1], to coupling with a potential flow solver of some kind to account for the non-uniform inflow at the propeller plane [2,3].

Fully discretized propeller computations in ships are reported in the literature. Lübke [4] computed the container KCS under self-propelled conditions using the commercial code CFX. Pankajakshan et al. [5] performed RANS calculations of the radio-controlled submarine ONR Body 1, but no free surface was modeled in this case. Venkatesan and Clark [6] also performed computations of the ONR Body 1, simulating a horizontal overshoot maneuver with a sliding mesh to model the rotation of the propeller and deforming grids to simulate deflection of the control surfaces. Turn and zigzag maneuvers for a KVLCC1 tanker with moving rudder and rotating propeller have been reported by Carrica and Stern [7] using overset grids and solving for the free surface. The most comprehensive self-propulsion computations to date have been performed by Carrica et al. [8], who studied a tanker (KVLCC1) a containership (KCS) and a surface combatant (ONR Tumblehome). All those computations were performed at model scale, though KCS is extrapolated to ship point using ITTC procedures.

CFD computations of ships are typically performed in model scale due to the lack of experimental results in full scale, and the added complexity of running codes at very high Reynolds numbers. Since experiments are simpler to perform and control in model scale, experimental data is available in model scale experiments against which CFD computations can be compared and validated. For self propulsion computations at ship point the ITTC provides of a procedure that allows to perform experiments and/or CFD in model scale and to extrapolate these results to full scale. The self-propulsion computations at model scale for KCS presented by Carrica et al. [8] follow these procedures, using the data from the Tokyo 2005 CFD Workshop [9] for comparison.

The work presented in this paper performs similar self-propulsion calculations on the KRISO container ship KCS presented by Carrica et al. [8] but at full scale. Since no open water characteristics are available for the propeller, an open water test at full scale is simulated. With the propeller open water results, the computation of a towed model and the self-propulsion computation at full scale the self-propulsion factors are obtained. These are compared against the CFD data obtained at model scale, which has extensively been validated in [8]. It is desirable in CFD computations to perform an uncertainty analysis (UA) to evaluate the accuracy of the results. The main component of the UA is a grid convergence study, usually studying systematically coarsened grids from the test grid. Coarsening is not possible in this case due to overlapping constraints in the overset scheme (coarser grids will not have

* Corresponding author. Tel.: +1 319 335 6381.
*E-mail address:* pablo-carrica@uiowa.edu (P.M. Carrica).

0045-7930/$ - see front matter © 2011 Elsevier Ltd. All rights reserved.
doi:10.1016/j.compfluid.2011.07.005

enough overlapping between grid blocks and thus will fail to find donors), and refining is prohibitively expensive given the size of the current problem. For these reason an uncertainty analysis was not performed. However, the accuracy of these CFD computations is established as best as possible by an exhaustive comparison with the available experimental data.

A method to estimate the input roughness for the wall function models is presented. Differences between model scale computations at ship point and full scale simulations are identified and discussed and some final conclusions are provided.

## 2. Modeling

The computations are performed with CFDShip-Iowa v4.5. CFDShip-Iowa employs a single-phase level set approach [10] to solve the viscous flow with free surface, using a RANS approach for turbulence modeling based on a blended $k-\omega/k-\varepsilon$ model [11] with SST. The code has DES capabilities as well [12] and wall functions are implemented for full scale simulations [13]. Multi-block structured grids with dynamic overset capabilities allow handling of complex geometries and large-amplitude motions, as described in [14]. The overset interpolations are performed at run time using the code SUGGAR [15], and the surface weights to compute forces and moments as a preprocessing step use Usurp [16]. Either Projection [17] or PISO [18] methods are used for pressure–velocity coupling and PETSc [19] is used to solve the resulting pressure Poisson equation. A PI speed controller is used to act on the propeller RPS to achieve the target speed. The velocity error is defined as the difference between the instantaneous ship speed and target speed

$$e_U = U_{\text{ship}} - U_{\text{target}} \tag{1}$$

and with the PI controller the instantaneous RPS is computed as

$$n = Pe_U + I\int_0^t e_U\, dt \tag{2}$$

where $P$ and $I$ are the proportional and integral constants of the controller. For details refer to [20].

Unless otherwise specified all variables are non-dimensionalized using a reference velocity $U_0$ taken to be the ship service speed and a length scale taken to be the length between perpendiculars $L_{PP}$. Then two dimensionless numbers define the simulation conditions: the Reynolds number $\text{Re} = U_0L_{PP}/\nu$ and the Froude number $Fr = U_0/\sqrt{gL_{PP}}$, where $\nu$ is the kinematic viscosity of water and $g$ is the acceleration of gravity. With these definitions the dimensionless rotational speed, forces and torque are $n^* = nL_{PP}/U_0$, $F^* = F/(\rho U_0^2 L_{PP}^2)$ and $M^* = M/(\rho U_0^2 L_{PP}^3)$, respectively, where $\rho$ is the density of water. The asterisk superscript to identify the dimensionless variables will be omitted from now on.

### 2.1. Hull roughness and sand grain equivalent

Ship point experiments or computations at model scale require the addition of the Skin Friction Correction (SFC) in order to extrapolate the results to full scale. This is an additional force in the direction of motion that takes into account the fact that the resistance coefficient at full scale is lower than the one at model scale. By adding the SFC the propeller is made to work at the same load condition, i.e. same advance coefficient $J$, at which it would work at full scale. The SFC is estimated from

$$\begin{aligned}\text{SFC} &= \{(1+k)(C_{F0M} - C_{F0S}) - \Delta C_F\} \times \frac{1}{2}\rho U_{0M}^2 A_{WM} \\ &= \text{SFC}^* \times \frac{1}{2}\rho U_{0M}^2 A_{WM}\end{aligned} \tag{3}$$

where for KCS at $Fr = 0.26$, $C_{F0M} = 0.002832$ and $C_{F0S} = 0.001378$ are obtained from the ITTC 1957 frictional line, $C_{F0} = 0.075/(\log_{10}\text{Re} - 2)^2$, $U_{0M} = 2.197 m/s$ is the reference velocity and $A_{WM}$ is the static wetted area [9]. Subscripts $M$ and $S$ denote model and full scale quantities respectively. $k$ is the form factor and corrects for the fact that $C_{F0}$ actually is the friction coefficient for a flat plate. $\Delta C_F$ is the roughness allowance and depends not only on the Reynolds number but also on the ship hull roughness. This can be estimated from the correlation proposed in the 19th ITTC [21].

$$\Delta C_F = 0.044\left[(k_s/L_{PP})^{1/3} - 10\text{Re}^{-1/3}\right] + 1.25\times 10^{-4} \tag{4}$$

where $k_s$ is the surface roughness. As explained in [21] this roughness is obtained using an instrument called Hull Roughness Analyzer. The roughness of a surface is a property which value depends on the measurement techniques involved and the subsequent statistical analyses. In Eq. (4) the surface roughness is well defined by the experimental procedure using the hull roughness analyzer.

On the other hand, most CFD codes (and in particular CFDShip-Iowa v4.5) treat surface roughness by the use of wall functions. The implementation of wall functions in CFDShip Iowa is reported in [13]. In CFDShip Iowa the two-point wall function model is implemented, where the velocity at the first node away from the wall is computed from

$$\frac{U}{u_\tau} = \ln(y^+) + B - \Delta B \tag{5}$$

where $u_\tau = \sqrt{\tau_w/\rho}$ is the friction velocity, $\tau_w$ is the shear stress at the wall and $y^+ = u_\tau y/\nu$ is the wall distance $y$ non-dimensionalized with the friction velocity. The Von Karman constant is $\kappa = 0.41$ and $B = 5.1$. $\Delta B$ accounts for surface roughness and results in a downshift of the logarithmic layer region [22]

$$\Delta B = \frac{1}{\kappa}\ln(1+\varepsilon^+) - 3.5 \tag{6}$$

where $\varepsilon^+ = u_\tau\ \varepsilon/\nu$ with $\varepsilon$ the surface roughness. The fundamental problem now is that the surface roughness appearing in Eq. (4) is not the same as the one in Eq. (6). In Eq. (6) $\varepsilon$ is the average height of sand-grain roughness elements in contrast to the roughness $k_s$ in Eq. (4), which is defined by the hull roughness analyzer measurement technique. Hence, in order to perform a CFD computation with surface roughness it is first necessary to find the sand-grain equivalent roughness to the one reported in experiments using the hull roughness analyzer. This is of utmost importance since as it will be shown these two definitions for surface roughness may differ in almost one order of magnitude for the case being considered.

At the Tokyo 2005 CFD Workshop [9] self propulsion computations at model scale were carried out using $\text{SFC}^* = 1.3294\times 10^{-3}$. However, no surface roughness was reported with which this value was obtained. Still, as reported in the 19th ITTC [21], Eq. (4) was calibrated using experimental data for which hull roughness ranged from 144 μm to 211 μm. Hence, surface roughness was expected to fall in this range. The roughness allowance can be obtained from Eq. (3) using the values of $C_{F0}$ obtained with the ITTC 1957 frictional line, the SFC used in Tokyo 2005 and the experimental form factor of $k = 1.1$, resulting in $\Delta C_f = 2.7\times 10^{-4}$. Using the correlation in Eq. (4) results in a surface roughness $k_s = 208$ μm, as measured with a hull roughness analyzer. With this the friction coefficient is $C_{FS} = C_{F0S} + \Delta C_{FS} = 1.643\times 10^{-3}$. This friction coefficient is an estimate for the KCS hull and corresponds to a flat plate of length $L_{PP}$. From [23] the drag coefficient for a flat plate in fully rough regime is

$$C_D = \left(1.89 + 1.62\ \log_{10}\left(\frac{L_{PP}}{\varepsilon}\right)\right)^{-2.5} \tag{7}$$