Table of Contents

Edited /Updated	Approved
Specialist Committee on CFD in Marine Hydrodynamics of the 27th ITTC	27th ITTC 2014
Date 02/2014	Date 09/2014

Practical Guidelines for Ship Self-Propulsion CFD

1. OVERVIEW

These guidelines are written as a complement to ITTC guidelines 7.5-03-02-03 (2011), "Practical Guidelines for Ship CFD Applications," and procedures recommended herein should then be in accordance to those guidelines.

Computation of the self-propulsion factors using CFD essentially mimics the ITTC recommended procedures and guidelines "Testing and extrapolation methods, performance propulsion test," 7.5-02-03-01.1 (2002). CFD evaluation of self-propulsion factors needs essentially data from three different tests: resistance, self-propulsion and propeller open-water tests. The first two are always performed with CFD, while the last can either be performed with CFD or propeller models based on potential flow theory, for a full numerical evaluation of self-propulsion factors, or can use experimental open water curves if available. In addition, computations can be performed at model scale, full scale, or at model scale with extrapolation to full scale.

2. SELF-PROPULSION COMPUTATION

2.1 Propeller Implementation

2.1.1 Direct Propeller Computation

The self-propulsion computation requires use of a propeller to provide thrust. The propeller implementation offers several possibilities of different degrees of complexity and computational cost.

In general, direct computation of the propeller provides the highest resolution but also the highest computational cost and the most complexity in terms of grid generation and code capabilities. The CFD code used has to be capable of rotating the propeller grid, which is typically done using sliding or overset grids, or employing the less accurate Multiple Reference Frame (MRF) method. Grid regeneration has also been attempted.

If the propeller will be computed directly, then the grid has to be designed to properly compute the friction forces. The Reynolds number used for the grid design is defined using the 70% radius chord length of the propeller blade and the local speed at that location

$$Re = \rho\omega 0.7 R C(0.7R)/\mu \qquad (0)$$

where ρ and μ are the density and viscosity of the fluid respectively, ω is the propeller rotational speed, R is the propeller radius and $C_{0.7}$ is the blade chord length at 70% radius location. Note that the viscosity of water varies with temperature and salinity and so proper values should be used. To evaluate C_f, as needed to compute the wall spacing needed for the desired y^+ as required for direct integration to the wall or use of wall functions, the blade chord length at 70% radius location should also be used. Since the precise propeller rotational speed is not known until a self-propulsion computation is performed, the design of the grid may result in an iterative process to determine the appropriate wall grid spacing. In most cases this can be avoided using an estimate of the propeller rotational speed at self-propulsion.

The time step is selected such that the propeller advances between 0.5 and 2 degrees per time step, with the higher number appropriate for implicit higher-order numerical integration schemes and the lower typical for explicit solvers. This time step, however, may be restricted by the CFL condition or by the implementation of the rotating propeller/moving grid algorithm and may result in a smaller time step than recommended herein.

2.1.2 Propeller Models

The alternative to direct computation is to model the propeller with some kind coupling between a propeller code and the CFD code. The propeller code receives wake velocities from the CFD code and returns volume forces on the fluid and shaft forces that are taken by the CFD code to accelerate the flow and propel the ship. As expected, increased complexity in the propeller code and the coupling strategy will lead to better results but more cost.

When the propeller is modeled the grid design may need to accommodate specific requirements for the coupling between the propeller code and the CFD code. These requirements are implementation-dependent and thus the best practices will depend on the specific software package used. As a general guideline, a grid fine enough to discretize the propeller radius in at least 10 elements, the circumferential direction in no less than 35 elements per quadrant and the axial direction in 10 elements covering the propeller thickness.

The selection of the time step in this case will be typically determined by the ship CFD computation, so the guidelines for ship CFD can be used.

2.2 Self-propulsion Computation

The grid design for the ship follows the same guidelines as in 7.5-03-02-03 (2011), as well as all other aspects related to a ship computation.

The computation can be performed in model or full scale, at even keel or free to sink and trim, as in experimental procedures, and even free to roll, yaw and sway for non-symmetric ships, though this is hardly ever done in experiments.

If the computation is performed in model scale and results will be extrapolated to full scale, as done in model scale experiments of self-propulsion, then a skin friction correction factor has to be added to the thrust to account for the reduction of the resistance due to friction in full scale respect to the model scale results,

$$F_0 = \left\{ (1+k)\left(C_{F,M} - C_{F,S}\right) - \Delta C_F \right\} \frac{1}{2} \rho S U^2$$

(0)

where the friction factors at model and full scale $C_{F,M}$ and $C_{F,S}$ are obtained from the ITTC 1957 model-ship correlation line, $C_F = \dfrac{0.075}{(log_{10}Re - 2)^2}$, U is the reference velocity and S is the static wetted area. k is the form factor and corrects for the fact that C_F is actually the friction coefficient for a flat plate. Note that the form factor is highly dependent on the Reynolds number and must be computed appropriately, see Practical Guidelines $7.5 - 03 - 02 - 04$ for Ship Resistance CFD. ΔC_F is the roughness allowance and depends on the Reynolds number and on the ship hull roughness. This can be estimated from the correlation proposed in the 19th ITTC

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

(0)

where k_s is the surface roughness as obtained by a Hull Roughness Analyzer. Alternatively the ITTC recommended value $k_s = 150 \times 10^{-6} \, m$ can be used. It is important to note that k_s is not the roughness used in CFD computations at full scale, in most cases based on wall functions. See Castro et al. (2011) for discussion, and comments on surface roughness in 7.5-03-02-03 (2011).

The self-propulsion computation requires finding the point at which ship resistance and propeller thrust are in equilibrium. Two approaches are common to achieve this goal. The first is to use a controller to change the propeller rotational speed to achieve the target speed or the balance between thrust and resistance for constant advance speed. This requires implementation of a controller and, if direct computation of the propeller is used, a dynamic change of the grid position, which precludes the use of CFD implementations that require pre-generated grids at different propeller positions. The second approach computes the imbalance between resistance and thrust for different propeller rotational speeds at the target ship speed and use iteration to find the self-propulsion point.

A third approach involves performing computations at different ship speeds and propeller rotational speeds and obtain load and resistance curves that are used to obtain the self-propulsion parameters.

2.2.1 Use of a Controller

A controller modifies the rotational speed of the propeller to minimize the error in either speed or resistance/thrust imbalance. A typical proportional-integral controller used to this end is implemented as

$$n = P\,e + I \int_0^t e\,dt \qquad (0)$$

where n is the propeller rotational speed in revolutions per second (RPS), e is the error and P and I are the proportional and integral constants. Choice of these constants depends on the definition of the error as described below.

If the code has a six degree of freedom (6DoF) capability, then the error is the difference between the ship's speed V and the target speed V_{target}, and the propeller rotational speed is changed by the controller to achieve the target speed

$$e = V_{target} - V \qquad (0)$$

When the prescribed target speed is achieved and stable the ship is in self-propulsion mode at that target speed. All results of the CFD computation can be used to obtain self-propulsion factors.

If the CFD code does not have a 6DoF capability a controller can still be applied using the imbalance in resistance F_x and thrust force T to define the error as

$$e = F_x - T. \qquad (0)$$

The evaluation of the optimal P and I constants requires some computation. If the constants are not properly chosen then overshoots and/or very slow convergence to the self-propulsion point may result. P and I can be chosen after simulation of the dynamic system behavior using a simple model with an approximate propeller thrust curve and ship resistance and mass properties, with the propeller rotational speed controlled by the controller. As an example for a 6DoF approach, the resistance can be estimated with

$$F_x = \frac{1}{2}\rho C_t S U^2 \qquad (0)$$

where C_t is an estimated resistance coefficient. The thrust and torque are obtained from

$$T = \rho K_t n^2 D^4 \qquad (0)$$

where D is the propeller diameter and K_t is an estimated thrust coefficient for the propeller near the self-propulsion point. Then the momentum equation

$$m\frac{dU}{dt} = F_x - T \qquad (0)$$

is solved in conjunction with the controller equation, evaluating different P and I constants to achieve the fastest convergence time to self propulsion.

2.2.2 Interpolation Method

In this case the self-propulsion point is obtained by trial and error using successive iterations. A typical method starts by performing a computation at the target speed with the propeller rotating at fixed RPS. The resulting resistance and thrust (and skin friction correction F_0 if computing in model scale with extrapolation to full scale) will be in general imbalanced. Then the RPS is adjusted appropriately to approach a better balance and a new run is performed. This new computation will result in a new imbalance in forces. Using the two imbalances and propeller rotational speeds an interpolation (or extrapolation) can be performed to predict the rotational speed that will zero the imbalance. A third computation is performed at this new propeller rotational speed, and if needed new interpolations and computations are performed until the imbalance of forces is acceptable. An acceptable force imbalance is less that 1% of the resistance, consistent with the accuracy of the best performed self-propulsion experiments.

3. TOWED COMPUTATION

A towed computation is needed to obtain the towed resistance. The towed computation needs to be performed under the same conditions of the self-propelled computation, but without the propeller. The resulting sinkage and trim may not match those of the self-propelled computations. See ITTC guideline 7.5-03-02-04, "Practical Guidelines for Ship Resistance CFD " (2014).

4. OPEN WATER CURVES

As mentioned earlier, a complete set of CFD computations to determine the self-propulsion factors is possible if an open water curve is obtained for the propeller. This is easily done if a direct simulation of the propeller is performed for the self-propulsion test. On the other hand, if a model is used for the propeller, it may be harder to perform computations of the open water curves since grids are not available, and use of experimental curves may be preferable, assuming these are available. This may introduce errors since propeller open water curves and CFD results are not matched. Alternatively, open water curves computed from potential flow codes can be used.

Direct computation of open water performance curves involves a series of computations of coefficients K_T, K_Q and η_0 for a propeller working in a uniform flow at a number of advance coefficients J. Similarly to open water

tests in a model basin, the propeller is arranged in pulling condition. Note that a new domain has to be defined and a new grid generated for open water computations. Due to the homogeneity of inflow the computation is usually carried out in steady mode with respect to a cylindrical coordinate system that is rotating at the same speed as propeller. Moreover, there is a blade-to-blade periodicity in the circumferential direction that can be utilized in defining the computational domain. Namely, it is sufficient to define a domain containing only 1/Z-sector of a full domain (where Z is the number of blades), using periodic boundary conditions on the azimuthal faces. The domain boundaries have to be placed sufficiently far away from the blade to avoid any boundary disturbance. Usually the inlet boundary is placed at least 2D (D=propeller diameter) upstream, and the outlet boundary at 4D downstream the propeller center. The outer boundary in the radial direction is placed at 4D or larger. Boundary conditions used for an open water computation are similar to those for a resistance calculation, see ITTC guidelines 7.5-03-02-03.

A hexahedral grid is preferable. If tetrahedral grid has to be used, a layer of prisms should be placed for the near wall region to resolve the blade boundary layer flow. The same guidelines in selection of turbulence model and discretization scheme as used for resistance calculations apply for open water computations.

The coefficient K_T, K_Q and η_0 are then calculated at a number of J values. The variation of J

can be achieved by keeping a constant inlet velocity while changing the rate of propeller revolution, or by keeping the rotation speed of the propeller and changing the advance velocity. K_T and K_Q are obtained by integration of pressure and viscous forces on propeller blades and the integrated torque along shaft. Note that the force on the hub is normally not included in the integration.

5. POST-PROCESSING

5.1 Computation of Self-Propulsion Factors

Separate evaluation of the forces and moments for both the ship and the propeller are needed to obtain the thrust and force imbalance. Details on this can be found in ITTC Performance and Propulsion Recommended Procedures 7.5-02-03-01.4 (1999). The thrust deduction factor is obtained from

$$t = \frac{T - R_T}{T} \qquad (0)$$

where R_T is the towed resistance. The advance velocity is usually obtained using the thrust identity method or the torque identity method. In the thrust identity method the thrust at self-propulsion is equaled with the thrust of the open water curve and the operating advance coefficient is obtained, $J_T = V_A / (nD)$, from which the advance velocity is obtained knowing the self-propulsion RPS. In the torque identity

method the torque at self-propulsion is equaled with the open water curve torque and the advance coefficient J_Q is obtained. Notice the use of subscripts T and Q for quantities obtained with the thrust or torque identity methods, respectively. The wake fraction is then computed as

$$w_{T,T} = 1 - \frac{J_T D n}{V} \quad \text{or} \quad w_{T,Q} = 1 - \frac{J_Q D n}{V} \quad (0)$$

The relative rotative efficiency of the ship propeller respect to the open water propeller is computed from

$$\eta_R = \frac{K_{Q,OW}}{K_Q} \quad \text{or} \quad \eta_R = \frac{K_T}{K_{T,OW}} \quad (0)$$

and the hull efficiency, relating resistance power to power at the advance velocity, is

$$\eta_H = \frac{R_T V}{T V_A} = \frac{1-t}{1-W_T} \quad (0)$$

which enables computation of the behind the hull efficiency, $\eta_B = \eta_R \eta_0$, and the propulsive efficiency $\eta_D = \eta_H \eta_B$.

Finally, the total efficiency can then be computed if the shafting efficiency η_S is known (typically dependent on lubrication and alignment)

$$\eta_T = \eta_D \eta_s. \quad (0)$$

6. VERIFICATION AND VALIDATION

No specific guidelines exist for verification and validation of ship self-propulsion computations. The V&V methodologies presented in the ITTC Recommended Procedures and Guidelines, Practical Guidelines for Ship CFD Applications, 7.5-03-02-03, can be applied to integral quantities such as propeller RPS, propeller thrust and torque and in general to all self-propulsion factors.

7. USEFUL REFRENCES

19th ITTC Report of the Powering Performance Committee. 19th International Towing Tank Conference, Madrid, Spain (1990).

ITTC Recommended Procedures and Guidelines. Practical Guidelines for Ship Resistance CFD. Report 7.5-03-02-04 (2014).

ITTC Recommended Procedures and Guidelines. Testing and extrapolation methods, performance propulsion test. Report 7.5-02-03-01.1 (2002).

ITTC Recommended Procedures and Guidelines. Performance, Propulsion 1978 ITTC Performance Prediction Method. Report 7.5-02-03-01.4 (1999).

ITTC Recommended Procedures and Guidelines. Practical Guidelines for Ship CFD Applications. Report 7.5-03-02-03 (2014).

Castro, A., Carrica, P.M., Stern F. "Full scale self-propulsion computations using discretized propeller for the KRISO container ship KCS," Comput. Fluids, Vol. 51, pp. 35-47 (2011).

Hino T, Editor. CFD Workshop Tokyo 2005, National Maritime Research Institute, Japan (2005)