Table of Contents

1. OVERVIEW 2
2. COMPUTATIONAL PROCEDURE 2
   2.1 Preliminaries 2
   2.2 CFD Code and Computer 3
   2.3 Ship Geometry, Computational Domain, and Boundary Conditions 3
   2.4 Computational Grid 4
   2.5 Discretization Schemes and Solution Algorithms 5
   2.6 Turbulence Modeling 6
   2.7 Convergence of RANSE Solutions 6
3. ANALYSIS OF RESULTS 6
   3.1 Resistance 6
   3.2 Effective Power Estimation 7
   3.3 Visualization 7
4. UNCERTAINTY ANALYSIS 7
5. REFERENCES 7
6. A TUTORIAL FROM THE G2010 WORKSHOP 8

Practical Guidelines for Ship Resistance CFD

1. OVERVIEW

These guidelines are written as a complement to the ITTC guidelines 7.5-03-02-03 (2014), "Practical Guidelines for Ship CFD Applications", and the procedures recommended herein are intended to be in accordance to those guidelines.

The ultimate goal of resistance computation is to determine the horsepower of a ship's engine that meets the ship's speed requirement. Resistance tests in towing tanks are conventionally carried out for a scaled ship model in calm water without propulsors. Resistance computations are frequently done to replicate the towing tank experiments. Resistance of the full-scale ship can be estimated by extrapolating the model-scale CFD result using the ITTC procedure. CFD can directly compute resistance of a full-scale ship, although it is computationally more challenging due to its much higher Reynolds number (Re ~ 10^9). The ability of CFD to compute both model- and full-scale resistance offers the community a new avenue to study model-ship correlations numerically.

Computation to determine the ship resistance yields, in addition, a large amount of flow data such as velocity, turbulence and pressure fields.

The following sections give a walk-through on various practical issues that need to be considered in resistance computations. Whenever possible, attempts are made to offer some practical recipes drawn from the best practices. Being written with computations using Reynolds-averaged Navier-Stokes equations (RANSE) in mind, these guidelines are applicable to both surface ships and underwater vehicles.

2. COMPUTATIONAL PROCEDURE

2.1 Preliminaries

In physical experiments or CFD computations alike, Reynolds and Froude numbers are two main dimensionless parameters that are relevant to viscous/turbulent free-surface flows around a ship. They are defined by:

$$\mathrm{Re} = \frac{\rho U L_{pp}}{\mu} \qquad (0)$$

$$\mathrm{Fr} = \frac{V}{\sqrt{g L_{pp}}} \qquad (0)$$

where ρ and μ are the density and viscosity of the fluid, respectively, V the ship speed, *L*pp the

length between perpendiculars of the ship, and g the gravity acceleration.

In rare cases where surface tension can play an important role, such as in a spilling breaker, the Weber number ($\rho U^2 L / \sigma$) should be considered as well.

These dimensionless numbers provide measures of relative importance of viscous, inertial, surface tension forces. The set-up of a CFD computation should ensure dynamic similarity with the ship's running condition in terms of these dimensionless numbers. Any combinations of ship length, speed, fluid viscosity, gravitational acceleration can be chosen as long as they match the relevant non-dimensional numbers.

2.2 CFD Code and Computer

Numerical prediction of ship's resistance requires a CFD code capable of computing high Reynolds number viscous/turbulent single- and two-phase flows around complex three-dimensional geometry. Offering good accuracy, reliability, and fast solution turnaround time, RANSE codes are the main workhorse for resistance computations, although LES and RANSE/LES hybrid approaches are conceivable.

In RANSE computations for a surface ship, sinkage and trim at a given speed – the running attitude of the ship - are not known a priori but parts of the solution. Thus, the RANS solver should have a capability to obtain a coupled solution of a two degrees-of-freedom (2-DOF) of rigid-body motion and the flow equations (RANSE).

With the typical grid size for ship applications in the range of several millions to hundreds of millions of grid points (or elements), high-performance computing on multi-processor or multicore machines has become a necessity. Many RANSE codes including in-house and commercial ones nowadays are ported on Linux/Unix clusters and multi-processor PCs. Some desktop workstations on the market are loaded with a sufficient large number of processors that give a reasonably short solution turnaround time for cases with grids several millions of grid points.

2.3 Ship Geometry, Computational Domain, and Boundary Conditions

Geometry of a ship is available these days in one of the formats widely supported by CAD packages including IGES, STEP, and STL files, to name a few. Care should be taken to ensure that the CAD geometry is exported with a sufficient numerical accuracy. CAD geometries often need to be repaired to make them water-tight and "de-featured" in cases where they have small open gaps and hydrodynamically insignificant features.

In general, the size of the computational domain should be taken sufficiently large. By the

general rule of thumb, the upstream, downstream and the lateral far-field boundaries are typically placed one ship length from the hull, and frequently more if the body is blunt or the Froude number is small. If comparing against experimental data the domain size is often decided to match the width and depth of the towing tank in which the resistance was measured. For unconventional geometries or conditions the appropriateness of the boundary conditions should be checked to determine if the boundaries are far enough. This is done, for example, by checking that the pressure gradient is actually approaching zero close to the boundaries if that boundary condition was used. Alternatively, checks can be performed by increasing the size of the domain and verifying that the solution in resistance converges.

Port-starboard symmetry of the flow can be taken advantage of to compute only a symmetric half of the full domain using symmetry boundary condition on ship's centerplane, which reduces the grid size by half. However, if computer resources are available, it is recommended to perform simulations of the whole domain to avoid loss of physics and generation of artifacts that may arise when a symmetry condition is imposed. Resistance computation for a surface ship can also be carried out with a "double-model" when the wave-making resistance is negligibly small or when one wants to determine the form-factor (k). In the double-model of a surface ship, the still-water plane is replaced by a plane of symmetry.

For boundary conditions, the solution variables such as velocity and turbulent quantities are normally fixed with known values on the upstream inlet and the far-field boundaries, whole zero-gradient conditions are applied on other boundaries.

For wall boundary conditions, one has two options to choose from:

- Near-wall-resolving approach in which the RANS equations are solved all the way down to viscous sublayer with no-slip velocity applied at the wall, or
- Wall function approach in which the viscosity-affected near-wall region is skipped with the aid of wall functions.

2.4 Computational Grid

In case of using a finite-volume solver permitting arbitrary polyhedral grids, it should be kept in mind that the type of the control volumes adopted (e.g., hexahedron, tetrahedron, wedge, pyramid) significantly affects numerical accuracy of resistance predictions. In any case, near-wall region including the boundary layer requires layered elements in the form of prisms. Hexahedral elements are known to provide best accuracy.

It is recommended that if the ship is symmetric, then the grid should be symmetric.

Ideally, the computational mesh should properly resolve all the significant features of

the flow including turbulent boundary layer, wake, vortices, and waves. It is imperative that mesh-dependency of the predicted resistance and flow-fields be investigated using systematic grid refinements, at least for one condition. See 7.5-03-02-03 (2014), "Practical Guidelines for Ship CFD Applications".

Regarding near-wall mesh resolution, there are two questions. One has to do with where (how close) to place the first grid point off the wall. The other question concerns how many grid points to put across the boundary layer. The general rule of thumb is:

- For near-wall resolving approach, the first grid point (or cell center) should be placed at $y^+ \sim 1.0$ or less, as required by most turbulence models. In some cases this distance may be even more restrictive. The user should check recommendations by the specific turbulence model and software used.
- For wall function approach, the first grid point should be in the log-law region, one should aim at the lower portion of the log-layer ($y^+ = 30 \sim 100$).

To properly resolve the hull boundary layer, especially its wake (velocity-defect) region, the grid should not be stretched too rapidly in the wall-normal direction. The old adage in the finite difference world still holds - the stretching ratio should be kept less than 1.2.

Free-surface flow computations using field equations such as volume fraction requires clustering grid points in a band wide enough to span the expected ship-generated waves, as well as enough grid points per resolved wavelength, as recommended in 7.5-03-02-03 (2014), "Practical Guidelines for Ship CFD Applications". A numerical experiment to quantify the effects of grid density near the interface on the resistance prediction is strongly recommended.

For prediction of sinkage and trim involving 2-DOF of ship motion, one can choose from single rigid-body grid, deforming grids, sliding grids, and dynamic overset-grids approaches as they are available in the RANSE solver.

2.5 Discretization Schemes and Solution Algorithms

One should employ, at minimum, second-order discretization schemes for advection terms to avoid excessive numerical diffusion plaguing the first-order upwind scheme that grossly overpredicts resistance and smears out the flow features. Higher-order methods can also be employed.

In cases where the volume fraction equation is solved to resolve free-surface, one should employ a 'compressive' advection scheme for volume fraction that can minimize smearing of the air-water interface.

Most of RANSE computations to predict ship's resistance and flow-fields seek steady-

state solutions. When an unsteady RANSE solver is used to obtain steady-state solutions, the first-order temporal discretization scheme is sufficient. For time-dependent flows, one would need a second-order temporal scheme such as three-level backward Euler or Crank-Nicolson's scheme.

2.6 Turbulence Modeling

Turbulence modeling significantly affects the accuracy of RANSE predictions of ship's resistance and flow-fields. Largely being an equilibrium turbulent boundary layer on the bulk of hull surface, ship flows can still carry complex features of three-dimensional shear flows such as crossflow in the boundary layer and the ensuing vortices emanating from hull and appendages.

The k-ω family of linear eddy-viscosity models seems to be by far the most widely used ones. One can benefit from second-moment closure models such as explicit algebraic Reynolds stress models and differential Reynolds-stress models that have been shown to better capture crossflows in the hull boundary layer and streamwise vortices from hull appendages.

2.7 Convergence of RANSE Solutions

A reduction of the "residuals" by a few orders of magnitude is clearly an indication of converging solutions, but is not always reliable. To ensure that the solution reached an unmistakable convergence, one should monitor the solution variables at some locations and the integrated quantities like the forces and moments, and continue iterations until they don't change.

3. ANALYSIS OF RESULTS

3.1 Resistance

Results of CFD computed resistance usually consists of frictional (tangential) and pressure (normal) contributions. The total resistance coefficient per the ITTC standard non-dimensionalization can be computed using:

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

where C_T is the total resistance coefficient and R_T is the total resistance. S is the hydrostatic wetted surface area of the ship.

The form factor, which is required for extrapolation of model-scale to full-scale result, can be computed from the resistance obtained from a CFD computation with flat free surface simulated with symmetry boundary conditions at the proper Reynolds number. Then

$$1+k = \frac{C_T}{C_F} \qquad (0)$$

where C_F is the ITTC -1957 Model-Ship correlation line. This is comparable to a double-

model wind tunnel experimental procedure to obtain *k*. For further discussion on how the form factor can be obtained see ITTC recommended procedure 7.5-02-02-01, Resistance Test (2011).

If low Froude number simulation results are used, the procedure to estimate the form factor is the same as tank test analysis based on ITTC recommended procedure 7.5-02-02-01, Resistance Test (2011). Notice that direct simulation of full scale resistance is possible with CFD, thus rendering unnecessary the computation of the form factor.

The residual resistance estimation can be done again in the same way as tank test analysis based on ITTC recommended procedure 7.5-02-02-01, Resistance Test (2011).

3.2 Effective Power Estimation

Effective powering estimation for a full scale ship is based on the ITTC recommended procedure 7.5-02-03-01.4, 1978 ITTC Power Prediction Method (2011).

3.3 Visualization

Several commercial software packages are available in the marketplace that can be used for visualizations of CFD results. A CFD result has a lot of information including surface and volume data. Among the surface and volume data relevant to ship's resistance are:

- Hull surface pressure (contour)
- Hull skin-friction (contour)
- Limiting wall streamlines on ship hull
- Wave elevation around ship
- Contours of mean velocity components at a selected number of planes
- Contours of mean vorticity vector components at a selected number of planes.
- Iso-surface of Q-criteria (2nd invariant of velocity deformation tensor)

4. UNCERTAINTY ANALYSIS

The ITTC procedures 7.5-03-01-01 and 7.5-03-01-02 already provide "methodology and procedures for estimating the uncertainty in a simulation result".

5. REFERENCES

ITTC, 2014. Recommended Procedures and Guide-lines. "Practical Guidelines for Ship CFD Applications," Report 7.5-03-02-03.

ITTC, 2011. "Resistance Test," Report 7.5-02-02-01.

ITTC, 2011. "1978 ITTC Power Prediction Method," Report 7.5-02-03-01.4.

ITTC, 2008. "Uncertainty analysis in CFD, Verification and Validation Methodology and Procedures," Report 7.5-03-01-01.

ITTC, 1999. "Uncertainty analysis in CFD, Guidelines for RANS codes", No. 7.5-03-01-02.

Larsson, L., Stern, F., Visonneau, M. 2014. "Numerical Ship Hydrodynamics - An assessment of the Gothenburg 2010 Workshop," Springer, Netherlands.

6. A TUTORIAL FROM THE G2010 WORKSHOP

An example of the application of these guidelines is illustrated by one of the Gothenburg 2010 (G2010, Larsson et al. 2014) workshop test cases. The example chosen was Test Case 2.1, the KCS hull form without rudder and in calm water conditions with zero sinkage and trim. The Reynolds number was defined as $Re = 1.4 \times 10^7$ and the Froude number defined as $Fr = 0.26$. The length of the hull was $L_{pp} = 230m$ at full scale with a scale ratio $\lambda = 31.6$ for the model scale measurements, resulting in a model length of $7.278m$.

The target Froude number ($Fr = 0.26$) gives the free stream speed for the model-scale test,

$$V_M = Fr\sqrt{\frac{gL_{pp}}{\lambda}} = 0.26\sqrt{9.81 \times 230 / 31.6} = \\ = 2.196\,m/s$$

The wave-length λ_W at this Froude number for the model scale geometry is given by:

$$\lambda_W = \frac{2\pi Fr^2 L_{pp}}{\lambda} = 3.0915m$$

From the Reynolds number, the skin-friction coefficient can be estimated as:

$$C_f = \frac{0.075}{\left(log_{10}\left(Re\right) - 2\right)^2} = 0.00283$$

The first grid point giving $y^+ = 1$ is:

$$y = \frac{y^+ L_{pp} / \lambda}{Re\sqrt{C_f / 2}} = 1.38 \times 10^{-5}\,m$$

The first grid point at $y^+ = 30$ is

$$y = \frac{30\,L_{pp} / \lambda}{Re\sqrt{C_f / 2}} = 4.14 \times 10^{-4}\,m$$

The CAD geometry was provided in an IGES file. The domain size used by the majority of the flow calculations carried out for the G2010 Test Cases 2.1 workshop had an upstream boundary of approximately L_{pp} from the bow, a downstream boundary of approximately $2L_{pp}$ from the stern, a side boundary L_{pp} from the plane of symmetry and a bottom boundary of L_{pp} from the keel, which is a little smaller than suggested earlier. A number of different positions were used for the top boundary, some methods computed the air flow above the ship hull and had a boundary of up to $0.5 \sim 1 L_{pp}$ from the keel and other contributors defined the

top boundary at the deck at $0.025L_{pp}$. The choice of top boundary position was dependent on details of the chosen boundary condition. from the stern, a side boundary from the plane of symmetry and a bottom boundary of $0.5\sim1L_{pp}$ from the keel, which is a little smaller than suggested earlier. A number of different positions were used for the top boundary, some methods computed the air flow above the ship hull and had a boundary of up to $0.5\sim1L_{pp}$ from the keel and other contributors defined the top boundary at the deck at $0.025L_{pp}$. The choice of top boundary position was depend-ent on details of the chosen boundary condition.

The majority of contributions to the G2010 workshop Test case 2.1 used hexahedral elements with expansion-ratio between 1.2 and 1.5 in the boundary layer. This expansion ratio is apparently a little too large per the guidelines presented in an earlier section, and could have affected resistance predictions. The majority of the workshop participants used hexahedral grids, although some contributors used unstructured grids with prisms and tetrahedral cells.

The number of cells per wavelength of at least 40 points per wavelength requires a maximum spacing in the axial direction of $3.095/40=0.0773m$. Almost all of the contributors went with this streamwise grid reso-lution.

Nearly all contributors used a variant of the $k-\omega$ family of models with some con-tributors using algebraic or differential Reynolds stress models. The majority of contributors used the near-wall-resolving approach with $y^+=1$, while some others employed wall functions.

The choices of numerical schemes made by the workshop participants are largely con-sistent with the guidelines presented in the earlier sections.