



**ITTC – Recommended
Procedures and Guidelines**

7.5–03
–03–02
Page 1 of 9

**Practical Guidelines for RANS Cal-
culation of Nominal Wakes**

Effective Date
2014

Revision
00

Table of Contents

1. OVERVIEW	2	3.2 Numerical Concerns	6
2. CHOICE OF MODEL OR FULL SCALE.....	2	3.2.1 Free Surface and Sinkage and Trim Effects	6
3. NOMINAL WAKE IN MODEL SCALE.....	3	3.2.2 Turbulence Model.....	6
3.1 Pre-processing.....	3	3.2.3 Discretization Scheme	7
3.1.1 Geometry.....	3	3.2.4 Convergence	7
3.1.2 Computational Domain and Boundary Conditions	3	3.3 Post-processing.....	7
3.1.3 Grid Generation	4	3.4 Verification and Validation	9
		4. REFERENCES	9

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



Practical Guidelines for RANS Calculation of Nominal Wakes

1. OVERVIEW

The present guideline is written as a complement to ITTC guideline 7.5-03-02-03 (2014), “Practical Guidelines for Ship CFD Applications” and the procedures recommended herein should be in accordance with that guideline.

Nominal wake refers to the flow field in the propeller plane at ship stern, when the ship is towed in the absence of propeller. Nominal wake is needed for propeller design, optimization of hull form (stern) and construction of wake wire screen for cavitation tests in small-sized cavitation tunnels where a dummy model is used instead of a ship model. In parallel to the traditional method of measuring wake in a towing tank, nominal wake can be computationally determined today with RANS methods. The computations can be performed at model scale, full scale, or at model scale with extrapolation to full scale wake by means of semi-empirical wake scaling methods.

Computational results of wake should be presented in the same manner as proposed by the ITTC recommended procedures and guidelines 7.5-02-03-02.4 (2011), “Nominal Wake Measurement by a 5-Hole Pitot Tube”.

2. CHOICE OF MODEL OR FULL SCALE

Most measured wake data is obtained on small scale models which have lower Reynolds numbers than the full scale ship. This means that the boundary layer is thicker at model scale than at full scale; hence the wake fraction is higher and must be scaled using empirical approaches to the full scale value.

RANS calculations can be performed at either model or full scale. Where model test data is available the CFD calculations should be run at the same scale for validation of the numerical model. Full scale RANS calculations can be performed to obtain the full scale wake directly. Whenever applicable and feasible, full scale wake calculations were recommended by the 26th ITTC Specialist Committee on Scaling of Wake Fields (2011). However, due to the higher Reynolds numbers full scale calculations tend to be less stable and may be difficult to obtain. In this case the wake can be obtained by extrapolation of a model scale wake field using a semi-empirical wake scaling method as would be used for the measured wake in a model test.

A number of semi-empirical wake scaling methods were assessed by the 26th ITTC Specialist Committee on Scaling of Wake Field, none of the methods gave perfect agreement with the full scale CFD result (used as reference). Yet Sasajima method shows relatively better performance in the upper part of the wake field than the other methods, the committee thus recommended this method with an alert. Users are referred to the final report of the 26th ITTC Specialist Committee on Scaling of Wake Field (2011) for further detail about the pros and cons of all the assessed scaling methods.

The guideline focuses mainly on the wake prediction at model scale. As many steps involved for a model scale calculation are also valid for a full scale wake calculation, no separate chapter is written for the latter. Instead, only the issues that differ from a model scale calculation are highlighted at the end of each sub-section in *italic* for a full scale calculation.

3. NOMINAL WAKE IN MODEL SCALE

3.1 Pre-processing

3.1.1 Geometry

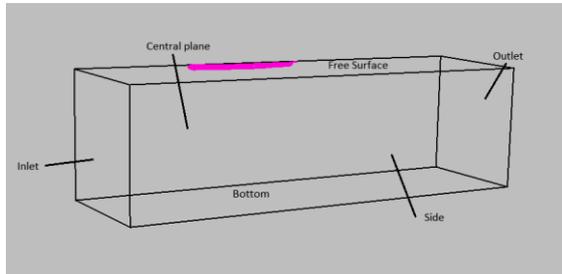
A smooth and well-defined hull geometry by means of a CAD surface description instead of 2D body plane is preferred. The geometry tolerance requirement for a numerical calculation is

usually higher than that for model manufacturing. If a visual inspection of hull surface reveals that there are wrinkles, visible gaps or overlapping surface patches, geometry repair and clean-up may be necessary prior to the grid generation step.

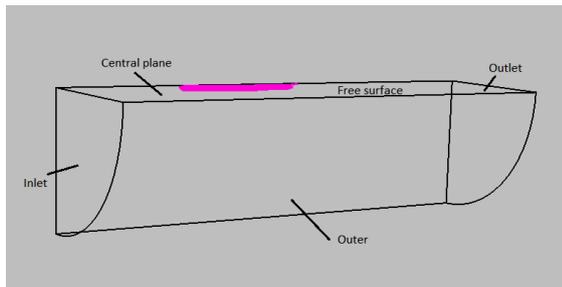
Bilge keels and stabilizer fins should be included in the geometry definition. Energy saving devices like ducts and pre-swirl stators that are placed upstream propellers must be included too. For open-shaft vessels the nominal wake is strongly influenced by the shaft and bracket arrangement. The geometry of these should also be included in the model.

3.1.2 Computational Domain and Boundary Conditions

Since the propeller is not present in the simulation and the ship is towed in a straight course, the flow is symmetrical with respect to the vertical central ($y = 0$) plane, it is frequently sufficient to compute half of the domain covering only the port (or starboard) side of the hull. The computation is normally carried out in steady mode. However, this may lead to loss of physics when transient flow occurs between starboard and port sides, mostly for DES or LES turbulence models, as well as artifacts caused by imposition of the symmetry boundary conditions. In such cases, whenever computational resources permit a full domain simulation is recommended.



(a)



(b)

Figure 1: Computational domains.

Topologically the computational domain is analogous to a 6-sided box. The domain is bounded by the hull surface, the central plane, the free-surface, an inlet and an outlet boundaries, a side and bottom boundary. In some cases, the side and bottom boundaries are replaced by a quarter of cylindrical outer boundary, as shown in Figure 1. The inlet, outlet, side and bottom boundaries should be placed sufficiently far away from the hull. In terms of ship length between two perpendiculars L_{pp} , the distances required for these boundaries are indicated as follows:

Inlet: $\geq 1.0L_{pp}$ upstream of FP,

Outlet: $\geq 1.0L_{pp}$ downstream of AP,

Side and bottom: $\geq 1.0L_{pp}$

where the side and bottom boundaries may need to extend further out for the case of blunt bodies.

The most commonly imposed boundary conditions are given in Table 1. Generally a symmetry condition is also assumed for the free surface (see §3.2.1).

Table 1: Boundary conditions

Boundary	Condition
Inlet	Constant velocity and turbulence quantities
Outlet	Constant static pressure
Hull	No-slip wall
Central plane	Symmetry
Top	Symmetry
Bottom	Symmetry
Side	Symmetry

3.1.3 Grid Generation

General guidance on grid generation is provided in the ITTC guideline 7.5-03-02-03 Practical Guidelines for Ship CFD Applications. For wake calculation hexahedral grids are the preferred grid type. If tetrahedral grids have to be used, at least one should avoid having tetrahedral grids within the boundary layer. Instead, generate several layers of prism grids in the inner boundary layer.

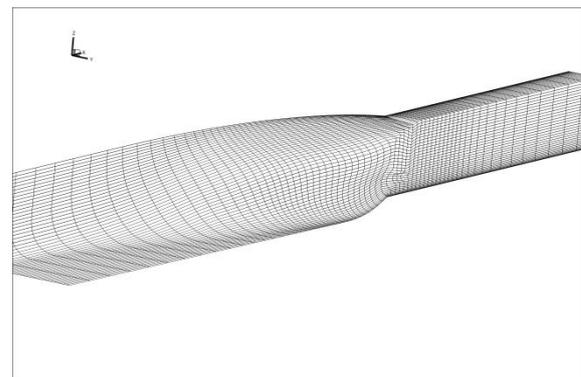
To resolve the development of boundary layer along ship and the wake flow at the propeller location, it is necessary to ensure the quality of grid and that the grid is sufficiently refined in the stern region.

Starting from the aft-shoulder of the hull, the grid in the longitudinal direction should be progressively refined towards the propeller plane and the refinement continues some distance (e.g. half propeller diameter) downstream. If the RANS solver is able to handle hanging nodes or overlapping grids, it may be worthwhile to make a further grid refinement at the stern. An example of progressive stretching of hull surface grid in the longitudinal direction and a local refinement at the stern bilge are shown in Figure 2(a) and (b), respectively.

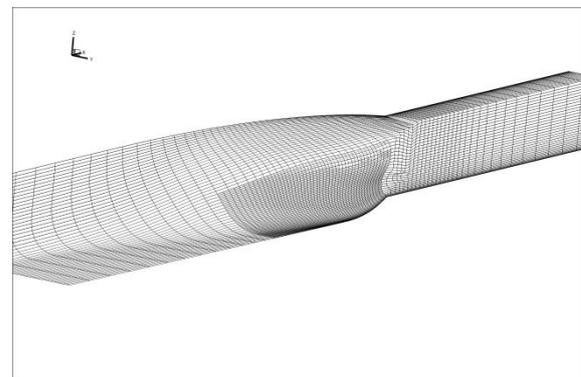
In the girthwise direction, enough grid nodes should be allocated in the region with rapid change of hull geometry, e.g. at the bilge. The grid lines growing out from the hull surface in the normal direction should be orthogonal to the girthwise grid lines as much as possible, as illustrated in Figure 3.

For grids in near-wall region, the number of grid nodes within the boundary layer is dependent on the turbulence model chosen and whether wall functions are used. If a wall-resolved turbulence model is adopted, the wall distance of the first grid point should be placed at $y^+ < 1$, with a growth ratio between two adjacent points no greater than 1.2. However, if a turbulence model

using wall functions is adopted, the first grid node should not be placed in the viscous sub-layer. Instead, it should be placed well within the logarithmic layer of the boundary, namely in the range, $30 < y^+ < 100$.



(a)



(b)

Figure 2: Example grid refinement at stern.

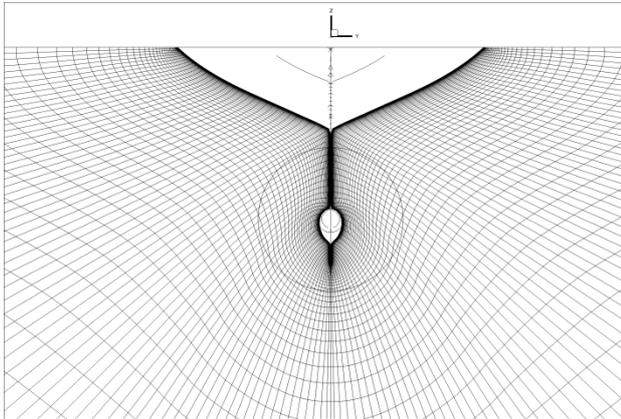


Figure 3: Grid orthogonality at a transverse cut.

Experience from the Gothenburg Workshop on Numerical Ship Hydrodynamics (Larsson et al. 2014) suggests that a 3~4 million grid on a half hull with a second order discretization scheme (without free surface effect included in the simulation) seems sufficient to make a good prediction of wake at model scale. For open-shaft arrangements, however, the need to resolve the additional detail of the brackets and shaft and their corresponding wakes may well exceed 20 million grid points.

For full scale wake, the higher Reynolds number requires a higher level of grid stretching from the wall if the target is $y^+ = 1$. This results in extremely high aspect ratios for the near-wall cells and a very large grid sizes. As a consequence, some solvers may encounter numerical instability or convergence problems due to extremely high stiffness in the system of equations. Under such circumstances, one may consider to

use a turbulence model that employs wall functions and generate a relatively coarser grid for the near-wall region with $y^+ > 30$.

3.2 Numerical Concerns

In addition to the issues addressed in the general guide line 7.5-03-02-03 (2014), the following aspects should be considered.

3.2.1 Free Surface and Sinkage and Trim Effects

The influence of the free surface and sinkage and trim on the flow at the propeller disc is normally assumed to be negligible for wake predictions for ships at low to moderate speeds. The common practice is to treat the free surface as a horizontal symmetry boundary, the ship model under such condition often being called a “double-body model”. For high-speed vessels where the stern wave may have an influence on the wake, it is then necessary to determine the free surface location as part of the solution to the wake prediction. The dynamic sinkage and trim have to be determined, either directly in the simulation as part of the solution (involving at least a 2-DOF motion simulation), obtained from towing tank tests, or from a separate simulation with a potential flow code.

3.2.2 Turbulence Model

Outcomes of the Gothenburg Workshop on Numerical Ship Hydrodynamics (Larsson et al.

2014) show that the turbulence model has a profound influence on the accuracy of local flow in stern region (under the condition that the other sources of errors, particularly the discretization and iterative errors, have been kept sufficiently low or properly controlled). The wake predicted by advanced turbulence models like RSM (Reynolds Stress Model), EARSM (Explicit Algebraic RSM) or SST $k - \omega$ with rotation correction show clearly better agreement with measured data than simpler turbulence models. This is particularly true for a full form ship where the developed bilge vortex with strong anisotropy on the Reynolds stresses significantly distorts the axial velocity at the propeller plane (appearing as a hook shaped wake iso-contour). These models should be considered at the first hand whenever possible; with the other simpler isotropic eddy-viscosity models (two-, one- or zero-equation model) as the second choice.

At full scale the bilge vortex on full form ship becomes less pronounced and there is practically no bilge vortex distortion (hook shape) present at propeller disc, the difference in the predicted wake pattern between the advanced turbulence models and linear eddy viscosity models tends to decrease, so there is a wider freedom in choosing turbulence models. On the other hand, one has to bear in mind that existing turbulence models are not sufficiently validated against full scale wake data yet.

3.2.3 Discretization Scheme

Any form of first order spatial discretization terms is to be avoided in a wake prediction. The second order upwind scheme is recommended. Discretization schemes higher than second order are also available in some RANS solvers and they provide better accuracy, yet at the price of longer computing time and solution instability.

3.2.4 Convergence

The convergence level of solutions can be assessed by monitoring the change of a few variables with the number of iterations, for example, the change of residuals. A common convergence criterion is “the decrease of scaled residuals by at least three orders of magnitude from their initial values”. Very often, the convergence is not solely judged by the residuals. The iterative history of an integrated quantity like total resistance or a local flow variable near the stern is also very useful information on judging the convergence of solution.

3.3 Post-processing

It is highly recommended that computational results are presented in accordance with the terminology and format proposed in the ITTC recommended procedures and guideline 7.5-02-03-02.4 (2011), “Nominal Wake Measurement by a 5-Hole Pitot Tube”.

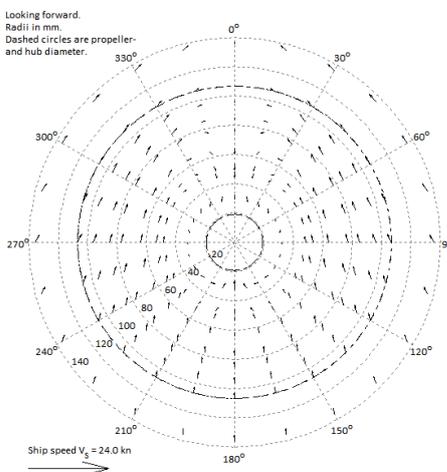
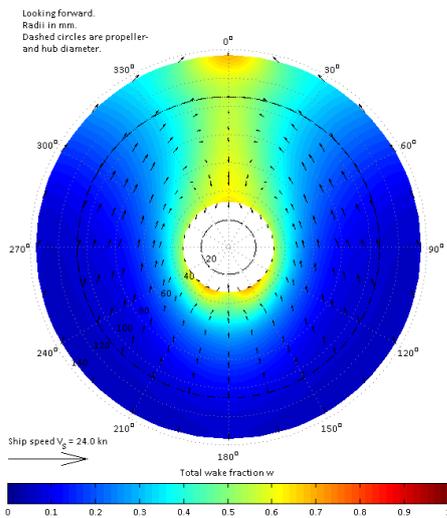
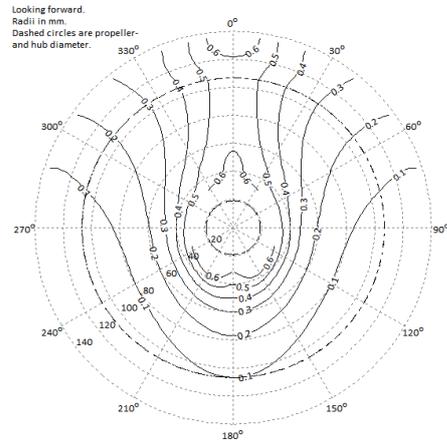


Figure 4: Iso-contours of wake fraction and
transverse velocity vectors

$r/R=0.600000$

ϕ	v/V_0	v_a/V_0	v_{tr}/V_0	v_t/V_0	v_r/V_0	w	w_a
0	0.41326	0.40808	0.0652	0.000002	-0.0652	0.58674	0.59192
15	0.48441	0.48278	0.03969	0.018818	-0.03494	0.51559	0.51722
30	0.59602	0.59353	0.05439	0.053711	0.008563	0.40398	0.40647
45	0.6832	0.67763	0.0871	0.084295	0.021929	0.3168	0.32237
60	0.74713	0.73998	0.1031	0.10228	0.012953	0.25287	0.26002
75	0.79685	0.78949	0.1081	0.107959	-0.00557	0.20315	0.21052
90	0.82886	0.82213	0.10536	0.102191	-0.02564	0.17114	0.17787
105	0.83831	0.83262	0.09756	0.087656	-0.04282	0.16169	0.16739
120	0.82892	0.82428	0.08757	0.068023	-0.05515	0.17108	0.17572
135	0.80682	0.80302	0.07825	0.047606	-0.06211	0.19318	0.19699
150	0.77453	0.77153	0.06806	0.027697	-0.06217	0.22547	0.22847
165	0.74588	0.74353	0.05923	0.011253	-0.05815	0.25412	0.25647
180	0.73091	0.72878	0.05565	-2.9E-05	-0.05565	0.2691	0.27122
195	0.74592	0.74356	0.05929	-0.01127	-0.05821	0.25408	0.25644
210	0.77458	0.77158	0.06809	-0.02767	-0.06222	0.22542	0.22842
225	0.80686	0.80306	0.07825	-0.04758	-0.06213	0.19314	0.19694
240	0.82894	0.82431	0.08755	-0.068	-0.05515	0.17106	0.17569
255	0.83832	0.83262	0.09754	-0.08764	-0.04282	0.16168	0.16738
270	0.82885	0.82213	0.10534	-0.10218	-0.02563	0.17115	0.17787
285	0.79684	0.78948	0.10808	-0.10794	-0.00557	0.20316	0.21053
300	0.74712	0.73997	0.10308	-0.10226	0.012961	0.25288	0.26003
315	0.68318	0.67761	0.08709	-0.08428	0.021937	0.31682	0.32239
330	0.59599	0.5935	0.05438	-0.0537	0.008573	0.40401	0.4065
345	0.48438	0.48275	0.03967	-0.01881	-0.03493	0.51562	0.51725

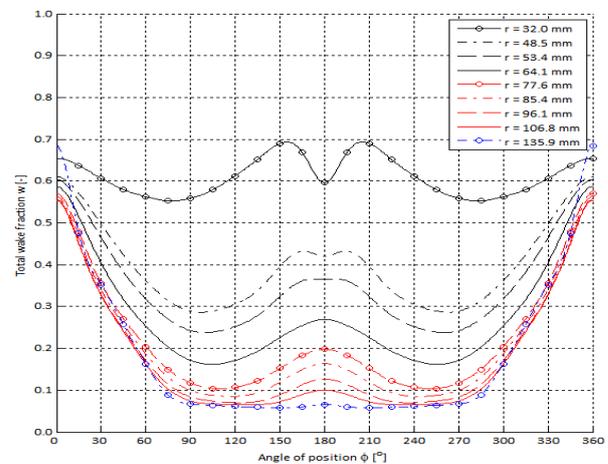


Figure 5: Wake components in tabular form (a,
top), and plotted against peripheral angles (b,
bottom).

Following this guideline, the wake fields are expressed by the axial (V_x), radial (V_r) and tangential (V_t) velocity component at the propeller plane. Note that all these components are normalized by the free stream velocity (or the towing speed). The results are usually presented as:

- a) Contour plot of iso-lines of wake fraction $(1 - V_x)$, including two circles to mark the radius of propeller hub and propeller tip, e.g. Figure 4.
- b) Transverse components (V_r, V_t) plotted as velocity vectors in propeller plane;
- c) The three velocity components are provided in tables at a number (often 10) of radius, or plotted as function of peripheral angles, e.g., Figure 5 (a), (b).
- d) Nominal wake fraction W_n by integration of axial velocity V_x over propeller disc.

Presenting the wake data in such a manner facilitates the use by the propeller designer.

3.4 Verification and Validation

A Verification and Validation (V&V) study is recommended to be carried out for each type of vessel to have an idea of the uncertainty level of computed wake. The study would give some information on what is the most suitable turbulence model of the particular RANS solver, and what is the best compromised grid size (between accuracy and computing cost) for wake prediction. This only needs to be done once in a while for the particular RANS solver until its next release (commercial software) or upgrade. A V&V analysis should also be performed every time a new type of vessel is analyzed. The V&V study can be performed according to the ITTC recommended procedure 7.5-03-01-01 or some other similar procedure.

At present, there are too few V&V studies performed for full scale wake predictions due to the obvious reason of lack of full scale wake data. The current practice is to perform some form of V&V study at model scale, then apply the same grid strategy, discretization schemes and turbulence model for the full scale case, assuming that the numerical models validated at model scale will perform equally well at full scale.

4. REFERENCES

- ITTC, 2011, “Nominal Wake Measurement by a 5-Hole Pitot Tube”, No. 7.5-02-03-02.4.
- ITTC, 2011, “Practical Guidelines for Ship CFD Applications”. No. 7.5-03-02-03.
- Hino T. Editor, 2005, “CFD Workshop Tokyo 2005”, National Maritime Research Institute, Japan.
- Larsson, L., Stern, F., Visonneau, M. 2014. “Numerical Ship Hydrodynamics - An assessment of the Gothenburg 2010 Workshop,” Springer, Netherlands.
- ITTC, 2011 “Final Report and Recommendations to the 26th ITTC”, The Specialist Committee on Scaling of Wake Field, Proceedings of 26th ITTC, Rio de Janeiro, Brazil.