Table of Contents

Edited /Updated	Approved
26th ITTC Specialist Committee on CFD in Marine Hydrodynamics	26th ITTC
Date 05/2011	Date 09/2011

Practical Guidelines for Ship CFD Applications

1. OVERVIEW

These guidelines are written assuming the use of surface capturing methods, the method found in most commercial and academic CFD packages. It also assumes that the solver is grid-based, as opposed to mesh-free methods.

We divide the CFD process into pre-processing, computation, and post-processing steps. The pre-processing step involves proper definition of the problem, grid generation and input setup to enable running of the computational code. The computation requires preparing the computer to run the problem, and running. The processing of the results of the computation to provide useful numbers and plots is called post-processing.

2. PRE-PROCESSING

2.1 Problem characterization

2.1.1 Resistance

Define the Reynolds and Froude numbers. These are given by:

$$Re = \rho\, U\, L_{PP} / \mu$$

$$Fr = U / \sqrt{g L_{pp}}$$

where ρ and μ are the density and viscosity of the fluid respectively, U is the ship speed, L_{PP} is the length between perpendiculars of the ship and g is the acceleration due to gravity. Note that the viscosity of water varies with temperature so that model scale tests carried out in water at 10-15° C is different to the full scale viscosity of deep sea-water at 15° C.

Estimate the expected free-surface wavelength and elevations. Estimate the distance to the wall where $y^+ <= 1$ for near wall boundary conditions and $30 < y^+ < 100$ for logarithmic wall functions. This estimation of the distance to the wall should be based on an estimation of the skin friction as a function of the Reynolds number. This distance is described in terms of the non-dimensional parameter y^+. This can be defined in terms of the Reynolds number of the required flow as follows:

$$y/L_{PP} = y^+ / (Re \sqrt{C_f/2})$$

$$C_F = 0.075/(\log_{10}Re - 2)^2$$

where y is the first required cell height and C_F is an estimate of the skin friction coefficient, based on the ITTC standard method. This gives an estimate of the skin friction coefficient at mid-ships.

Note that in the calculation above, C_F should be calculated using the full length of the ship which in the approach gives an approximation to mean value of the local C_F The ITTC model-ship correlation line should only be used in this estimate.

2.1.2 Wall function

To resolve the high velocity gradients in the inner part of a boundary layer, strong contraction of grid nodes is required towards solid surfaces . The cells that are thus occurring close to the wall have a high aspect ratio; typically for a ship at full scale they can be 1 meter long, 0.5 meter wide and only 10^{-3} millimeter high. This not only results in a high mesh count, but it also poses strong demands on the mathematics in the numerical solution procedure. This led to the introduction of so-called wall functions. For the laminar flow over an infinitely long flat plat at zero pressure gradient, the well-known Blasius solution exists, where the velocity profile is independent of the streamwise co-ordinate, when scaled appropriately. For turbulent flows such solutions only exist for the innermost part of the boundary layer. This gives the possibility to remove a significant part of the cells close to the wall (which also have the highest aspect ratio), and impose the velocity at the first grid node adjacent to the wall (the wall function). However, wall functions are based on two-dimensional flow, typically at zero pressure gradient, and it is well known that the validity of these analytical expression becomes less, or even disappears, with increasing adverse pressure gradients. Thus it cannot be expected that the wall function approach leads to reliable and accurate solutions near a ship stern, where the flow is strongly three-dimensional and running up against an adverse pressure gradient. Thus it is a trade-off between accuracy and computational effort.

Wall functions should be prevented if possible, and used with care when necessary.

2.1.3 Surface roughness

In ship viscous-flow computations and even in model testing the hull of a ship is considered to be hydrodynamically smooth. At full scale, however, during the operation of a ship the roughness increases due to use and fouling, thus increasing the frictional resistance. There are basically two methods to include roughness effects in RANS computations. Either through the adaption of wall functions (when used) or through the adaptation of the turbulence boundary conditions (for instance in the k-ω model). However, some major problems still remain. First the validity of the roughness model: to what extent does the equivalent sand-grain roughness that is typically used correctly adjust the velocity profile close to the wall? And second, how can the condition of the surface of a ship, sailing at any part of the world, be translated to an equivalent sand-grain roughness?

Surface roughness is still an active field of research, and no general guidelines can yet be given.

2.1.4 Incident waves

Define the wavelength, amplitude, and encounter frequency. For irregular waves define spectrum and proper parameters. Define the cut-off frequency (highest frequency to be included in the simulation), and determine the corresponding wavelength.

2.1.5 Motions

Estimate the frequencies and amplitudes of each of the motions to be simulated (surge, sway, heave, roll, pitch, yaw) and the amplitude of displacement of the ship.

2.1.6 Flow features

Define location, size and characteristic frequency of the flow features to be resolved (vortices, separation, flutter, etc.).

2.1.7 Region of influence

Estimate the extent of the domain to be simu-lated to minimize interaction of the boundary conditions with the simulation results.

2.2 Geometry creation and modification

The geometry is generally provided as surface definitions in an IGES file format. Alternative file formats may be also used provided care is taken to ensure that sufficient accuracy is maintained during the file transfer process. The accuracy of the geometry should be checked to ensure that the surface definitions are reasonably smooth and connect within a given tolerance. The tolerances for the geometry should be based on the length between perpendiculars. The required geometry tolerance also depends on the Reynolds number required for the flow calculations.

Generally, appropriate geometry tolerances are:

Scale	*Lpp*(m)	Re	Tol.(m)
Model	$1 < L_{PP} < 10$	10^6 - 10^7	10^{-5}
Interme-diate	$10 < L_{PP} < 50$	10^7 - 10^8	$5x10^{-5}$
Full	$50 < L_{PP} < 250$	10^8 - 10^9	10^{-4}

Due care and attention is required to resolve geometry features such as trailing edges that may be less than an order of magnitude larger than the geometry tolerance. Geometry features that are smaller than the geometric tolerance do not need to be resolved.

Additional geometry is required for the grid generation process. It is recommended that this geometry is produced within the grid generation package to the same geometric tolerances as the surfaces. The required geometry components are:

- Bounding surfaces of the grid domain surrounding the geometry
- Intersection curves between the appendages and the hull
- Intersection curves between the hull and appendages and the vertical plane of symmetry
- Additional curves may also be required to assist in defining key details of geometry

Due consideration to the position and orientation of the origin of the geometry should be taken depending on the type of problem to be solved as forces and moments will be obtained about this origin and velocity and rotation directions depend on the orientation. It is recommended that a consistent coordinate system be used within an organization. Imported CAD definitions should then be modified to conform to this system.

2.3 Grid generation

Details on the grid generation process will largely depend on the solver and the type of grids it can handle (Cartesian, structured multiblock, unstructured, overset, etc.) Here are some general guidelines that apply to most solvers.

2.3.1 Definition of the domain boundaries

Ship viscous-flow computations typically have three fixed boundaries: the ship surface, the symmetry plane and the (still) water surface. Furthermore three additional boundaries have to be defined in order to have a closed domain around the ship. Independent of the grid type used these will include an inlet, an outlet and an exterior boundary, where *approximate* boundary conditions have to be defined. These boundaries have to be placed sufficiently far from the ship to minimize the effect of the location of these boundaries on the solution. For the inlet and exterior boundary either the uniform (undisturbed) flow is usually imposed, and in that case the these boundaries should be located 1-2 L_{PP} away from the hull. Alternatively potential flow can be imposed at these boundaries, which enables a reduction of the domain size. At the outlet in general zero gradients for all unknowns are imposed.

In case free-surface boundary conditions are imposed on the water surface, the domain size has to be increased further. Preferably the Kelvin wedge does not intersect with the exterior boundary, to prevent wave reflection.

Unsteady methods often require a damping zone downstream, to prevent wave reflection from the outlet boundary. There the outlet has to be placed 3-5Lpp downstream.

2.3.2 Element type

Quadrilateral (2D, 4-sided) and hexahedral (3D, 6-sided) elements are the most popular element types supported by almost all CFD codes. Topological attributes of these elements – presence of opposing faces, and relative locations of cell centers and face centers – have been found to be beneficial to spatial accuracy of numerical solutions. In typical mappable, structured mesh-based solvers, the presence of stencils (e.g., *i*, *j*, *k* coordinates in the computational domain) readily accommodates high-order discretization schemes (e.g., 5th-order convection scheme) that can enhance spatial accuracy. They are also efficient in terms of usage of elements, since they can be clustered and/or stretched as needed to economically resolve the flow fields. The main downside of these structural mesh elements is that it is often very hard to generate high-quality structured meshes for complex geometry.

The “unstructured” mesh gives more flexibil-ity in the choice of element types, facilitating mesh generation for complex geometry. The majority of unstructured mesh-based CFD solvers allow use of arbitrary polyhedral elements such as quadrilaterals (2D), triangles (2D), hexahedra, tetrahedra, wedges, pyramids, prisms, to name a few, and combination of all them (hybrid unstructured mesh). In a typical unstructured mesh frequently adopted in ship hydrodynamics, hull boundary layer is discretized using a prism mesh grown out of triangular mesh on hull surface, and tetrahedral mesh is used elsewhere away from hull. Compared to typical structured meshes, meshing time can be dramatically reduced with unstructured meshes. Spatial accuracy for unstructured mesh elements such as triangles, tetrahedra, and pyramids can be lower than that for quadrilateral and hexahedral elements. With unstructured meshes, one usually need a far greater number of computational elements than structured meshes in order to achieve a comparable accuracy. Furthermore, spatial accuracy of the majority of unstructured mesh-based finite-volume solvers is limited to 2nd-order.

Which element types to use for a given problem really depends on many factors such as the solver (does your solver support unstructured mesh?), objective of the computation (do you need to resolve fine details of the flow?), and computer resource (do you have computers to run cases involving large meshes?). Here are general guidelines in choosing mesh and element types.

- For relatively simple configurations such as bare hulls, consider using a high-quality hexahedral mesh.
- For relatively simple configurations involving body-motion (free sinkage and trim), consider using overset grids if your

CFD solver can take them and run on them.

- For complex configurations such as fully-appended ships for which a high-quality structured mesh is difficult to generate, consider using an unstructured mesh, preferably a hybrid unstructured mesh.
- Avoid using tetrahedral mesh in boundary layers, near free surface, and in the regions where high resolution of flow-fields is required. Use hexahedral grids or prismatic grids instead.

2.3.3 Grid points

Grid points distributions are determined with consideration of the following points.

- Based on the availability of computer time and power, determine the size (total number of grid points) of the grid as well as the grid size required for previous similar problems. This should determine if sufficient resources are available to obtain reliable results.

- Design the grid blocks in such a way that they will be properly decomposed for efficient computation, avoiding the use of too many small blocks.

- Use no less than 40 grid points per wavelength on the free surface. In irregular waves use at least 20 grid points for the shortest wave length to resolve. The number of grid points per wavelength also depends on the order of accuracy of the numerical scheme so if 40 points are required for a 3rd/4th order method then 80 points are required for a 2nd order scheme to obtain the same accuracy (as provided by most commercial codes).

- comment on

(1) the minimum number of points per wavelength: for too short wavelength (small Froude number) it is probably pointless to try to catch the small waves: the wave resistance component is likely negligible and

(2) minimum grid density to capture flow details (e.g. wake at the propeller disk) or global forces (i.e. total resistance)

- Use no less than 20 grid points in the vertical direction where the free surface is expected

- Whenever possible use orthogonal grids to resolve a free surface

- For turbulence models integrating to the wall (Spalart-Allmaras, k-ω, etc.) locate the first grid point at a distance from the ship's wall such that $y^+ = 1$. If wall functions are used this distance can increase such that $30 < y^+ < 300$, depending on the wall function implementation.

- Whenever possible use hyperbolic grid generators to guarantee as much as possible an orthogonal grid near the wall.

- Grids orthogonal to the domain boundaries, where the boundary conditions are imposed, are recommended. For some boundary conditions, such as symmetry conditions and wall conditions this orthogonality condition may be mandatory for some solvers.

- Provide refinement where flow features of interest are expected, in accordance with the size of the feature to be simulated. Where the flow features of interest are not known beforehand, it is necessary to use an iterative process to establish the existence of the key flow features of a given geometry. This requires an initial flow solution to be obtained and examined and subsequent grid refinement provided around the flow features. This should be carried out until some measure of grid refinement index is satisfied to ensure that flow features are sufficiently resolved.

- If overset grids are used, check that overlap is sufficient for the number of fringes that are needed in your code and order of accuracy.

- Ensure sufficient resolution of high curvature geometry is provided, especially around leading and trailing edges. An appropriate grid structure can enable more efficient use of computing resources but at the expense of increased grid generation time and complexity.

- Check the grid quality to guarantee that all volumes are positive (positive Jacobian in structured grids), skewness and aspect ratio are acceptable, and that orthogonality is nearly satisfied in most places.

2.3.4 Grid topology

Grid topology is the mapping relation between the grid surface in physical (x, y, z) space and the computational (i, j, k) space in case of structured grid systems. In the single block grid around a ship hull, O-O or H-O topology is adopted in most cases, although C-O and H-H grids can be applied. In either topology, the grid lines in the girth-direction are O-type. The longitudinal grid lines in O-O grids wrap around a ship hull whereas those in H-O grids start from the inflow boundary and go through regions ahead of a ship, side of a hull and aft of a ship. Thus, when the total number of grid points and the number of grid points along the lines in the normal and in the girth directions are fixed, O-O grids can accommodate more grid points along a ship hull than H-O grids. Also, O-O grids can be adapted more easily to a blunt bow or a wide transom sterns which are typical in modern commercial ships. On the other hand, since the grid lines in the normal directions spread to the outer boundary in O-O grids, the grid resolution in the wake region and the region away from a hull tends to be lower than the H-O grids. Therefore choice of grid topologies should be based on the nature of ship hull geometry and on the consideration of which part or which feature of flow fields is more important than others.

2.3.5 Non conformal mesh

Non conformal grids may be required for highly complex geometries. This occurs when the level of detail of the geometry is increased and as more geometric entities such as propulsion shaft and bracket arrangements and bilge keel and roll stabilization systems are included in the geometry definition. These types of geometries are often required for wake flow analysis where it is important to capture small flow features. For these types of grids a non-conformal grid may be more appropriate where parts of the total grid do not fully connect. For these grids the flow solution algorithm must use special coding in order to interpolate between the non-connected grids. For some methods this interpolation scheme may be defined by 'interfaces' where the interpolation method is defined across grid boundary surfaces, for other methods the interpolation scheme is defined by 'overlaps' where the interpolation is defined across local grid volumes. For both of these types of interpolation schemes the formal order of accuracy is likely to be reduced, especially when there are large differences between the grid resolution and topology. However, this type of approach can be used to considerably simplify the grid generation process so that locally better quality grids can be produced around the various geometry components and assembled together to form a complete grid using the interpolation schemes.

Non conformal grids should be used with care, for example, for free-surface flows undesira-ble wave reflections can occur at the interfaces or overlap if the interpolation scheme is unable to resolve the change in grid correctly. This can be alleviated by ensuring that the local change in the grid across the non conformal region is minimized with similar grid resolution and spacing used for both grid regions. Non con-formal grids can also use different types to assist the grid generation process, for example a grid for a detailed rudder with skeg and end plates can be produced using a local prism/tetrahedral grid which is embedded inside a multiblock grid for the hull.

2.3.6 Expansion ratio and number of grid points in boundary layer

The number of points within the boundary layer is determined by the level of accuracy required and the turbulence model chosen. A near wall turbulence model resolving the laminar sub-layer needs at least 3 points inside it, which for a y+=2 results in an expansion ratio of 1.5, the largest acceptable. In most cases a y+=1 will be used with expansion ratios around 1.2. Wall functions start farther out in regions of smaller velocity gradients and can use large expansion ratios, as large as 1.5 for coarse grids. In fine grids integrating all the way to the wall the total number of points within the boundary layer can be very large, on the order of 100, while for coarse grids with wall functions less than 10 will suffice.

2.3.7 Grid skewness

Typically the 3x3 determinant for structured grids should be greater than 0.3, as a measure of the Jacobian and associated skewness. However, it may be necessary to have a few small cells where the 3x3 determinant is no better than 0.15. For these cases it may be necessary to use a smaller time step or increased under-relaxation in order to achieve converged results.

2.4 Boundary conditions

Choose boundary conditions that are compatible with the domain size chosen and problem approximations.

- Inlet conditions that are far from the field can impose pressure, velocity and free surface elevation as Dirichlet boundary conditions mimicking free flow. If the boundary is close to the ship Neumann boundary conditions (zero normal gradients) need to be imposed to relax the pressure and free surface.

- Far field boundary conditions can also be free flow if the boundary is far from the object (typically one ship length for Froude numbers larger than 0.2). If the boundary conditions are close or the Froude number is small then the ship will affect significantly the flow on the boundary, and Neumann conditions are preferred.

- Exit conditions usually are modeled with zero second derivative for velocities (zero traction) and zero gradient for pressure and free surface. This condition requires no inflow from this boundary, so it must be placed far downstream enough to guarantee this throughout the computation. Otherwise zero normal gradient conditions can be attempted.

- The bottom can be treated as a far field if deep water is being simulated.

Sides and bottom may need to be treated as moving boundaries if shallow waters or a narrow towing tank are being simulated.

- In case of free surface flow simulations, the so-called radiation conditions must be imposed on exit or far field boundaries in order to prevent wave reflection on boundaries. Simple way to implement no reflection condition is to damp waves which go through boundaries with numerical dissipation by use of large grid space near the boundaries or by explicitly adding artifical damping terms to the governing equations. This is often called ‘numerical beach‘ approach.

2.5 Choice of the time step

In explicit solvers the time step is chosen to satisfy the CFL condition or to resolve the flow features of interest, whatever results smaller. Usually the CFL condition is more demanding than the flow requirements. In implicit solvers the time step is decided by the flow features. As a rule of thumb:

- For waves, use at least 60 time steps per period for the shortest waves, or 100 time steps per period for regular waves.

- For other periodic phenomena (roll decay, vortex shedding, etc.) use at least 100 time steps per period.

- For complex unsteady phenomena, like wetted transom instabilities, use at least 20 time steps per period for the highest frequency to be resolved.

- For rotating propellers use at least 200 time steps per revolution.

- For standard pseudo-transient resistance computations, use Δt = 0.005 ~ 0.01 L/U. The choice of time step will also depend on the complexity of the turbulence model. For Reynolds stress turbulence models it is more appropriate to use Δt = 0.001 ~ 0.0025 L/U. This also requires a larger number of iterations to obtain reasonable convergence. In more unstable problems, as those with low Froude number, a smaller time step may be needed. Notice that naturally transient problems will not reach a steady-state solution.

2.6 Choice of convergence criteria

A number of convergence criteria should be defined and examined in order to ensure reliable convergence of solution.

At first hand the level of convergence should be assessed by the history of residual variations for the mass and momentum equations. Residuals indicate how far the present approximate solution is away from perfect conservation (balance) of mass and momentum. Thus the residual for a discretized equation is defined as the L_1-Norm of the imbalance between the left and the right hand side of that equation over all the computational cells. Usually in definition the residual is also scaled by a reference value. Sometimes L_2-Norm and L_∞-Norm are also used to define residual.

CFD users do not need to worry about the definition of residuals, as they are often predefined by code developers. Instead, attention should be paid to the selection of convergence criteria. The recommended criterion is “*the drop of scaled residuals by at least three orders of magnitude off their initial values*”. However if this criterion cannot be achieved due to complexity of the problem or oscillatory convergence is found, then other criteria can be used to assess the convergence of the globally integrated parameters, for example:

- Forces and moments acting on the hull
- Thrust and torque produced by the propulsion system
- Velocity and turbulence parameters in key region of the flow field (e.g. at propeller plane for nominal wake calculations)

2.7 Choice of free surface model

There are two major categories in free surface models. First one is an interface fitting approach in which a numerical grid is aligned to deformed free surface shape and the other is an interface capturing approach in which a free surface shape is defined as an iso-surface of a marker function and a grid does not to fit to a free surface. Choice of a free surface model is needed when a flow solver used offer both interface fitting and capturing models.

The interface fitting approach is more accurate and efficient than the capturing approach, since free surface boundary conditions can be applied in the exact free surface location. Therefore the interface fitting model may be selected whenever it is possible. However, it should be noted that re-gridding procedure is essential in the interface fitting method in order to keep the gridlines follow the deformation of free surface. This may cause severe distortion of gridlines even though the initial grid fitting to an undisturbed free surface has a good quality.

Difficulties in grid generation and/of re-gridding can be avoided in interface capturing approaches. Also, in the case that large deformation of free surface, such as overturning or breaking waves, is expected, interface capturing methods should be used. Since the capturing methods demand finer grid resolutions in the interface zones, grid generation requires more attentions. Choice of the level-set function method and volume-of-fluid method in the capturing approaches is little impact in the final solutions. Although details of numerical procedures are different from each other, well-examined flow solvers provide similar results in both models.

2.8 Choice of turbulence model

Turbulence modelling has been an important research topic over the last decades. A large number of models have been proposed, tested and applied, but no ‘universal’ model has been developed. Thus one is forced to choose the best model available for each specific application. The majority of turbulence models is based on the so-called Boussinesq hypothesis, which defines a turbulence or eddy viscosity (as opposed to the molecular viscosity) to account for the effect the turbulence motion has on the mean flow.

<u>Zero-equation, or algebraic models</u> express the eddy viscosity in terms of the mean flow variables and mean flow gradients without solving any additional equations. They are hardly ever used in ship hydrodynamics.

<u>One-equation models</u> solve one additional equation (i.e. in addition to the momentum and mass conservation equations) for the eddy viscosity. Regularly encountered in ship hydrodynamics are models by Menter and by Spalart-Allmaras. These models are sometimes extended with a correction for vortical flow, to improve wake field predictions.

<u>Two-equation models</u> solve two additional equations for the eddy viscosity, one for the turbulence kinetic energy (k), and one for its dissipation rate (typically ε or ω). These models have shown to be able to give accurate predic-

tions in ship hydrodynamics, especially certain versions of the k-ω model and are by far the most applied ones (80% of the submissions for the Gothenburg 2010 Workshop).

An important class of turbulence models, not based on the Boussinesq hypothesis, are the Reynolds-stress models, and versions thereof. Rather than introducing an eddy-viscosity, they aim to solve the equations for the six Reynolds stress components directly. Apart from that, additional equations have to be solved, since terms in these equations require modelling as well. Consequently Reynolds-stress models are more computationally intensive, and often less easy to converge, compared to the one or two-equation models. However, they contain more physics and can be expected to be more accurate than eddy-viscosity models.

A more recent development is Large Eddy Simulation (LES). Other than the turbulence models discussed above it does not average the Navier-Stokes equations in time, but filters them in space. This results in transient computations on extremely dense grids as they aim to resolve all turbulence motion to a very small scale. Detached Eddy Simulation (DES) is a hybrid method that tries to reduce the required computational effort by solving the (unsteady) RANS equation in the boundary layer and applying LES in the rest of the domain. However, the very high Reynolds numbers encountered in ship hydrodynamics prevents the application of both methods in practical design projects.

2.9 Choice of numerical scheme

In the majority of industrial CFD codes, diffusion terms in the governing equations are discretized using a second-order (central differencing) scheme by default. Thus, spatial accuracy is largely determined by discretization scheme used for convection terms.

The first-order upwind (FOU) scheme, offered in many commercial CFD codes often as a default scheme, is famously stable. However, it introduces an unacceptably large amount of numerical (false) diffusion - that is why it is so stable. Therefore, it should be avoided at all costs. However, the very robustness of the FOU scheme can be exploited to start up the solution. For example, the first 100 iterations (or time steps) during which the solution is most susceptible to numerical instability and divergence) can be run using the FOU. As the flow-fields start settling down, one can switch to a high-order scheme.

The majority of high-order convection discretization schemes in popular use today formally have a second-order of accuracy with an upwind bias. All these second-order upwind (SOU) schemes differ from one another in terms of the flux limiter used to suppress unphysical oscillations in the solutions. Still higher-order schemes such as 5th-order scheme exist. However, not all CFD solvers offer such higher-order schemes. Even if they are available, the lack of robustness often makes them less useful than claimed. The SOU scheme that is both reasonably accurate and robust, and for that reason is an industrial workhorse for convection discretization. The SOU scheme is therefore recommended for all convection-diffusion type of transport equations.

Volume-fraction equation requires a special care, inasmuch as the transported quantity is essentially a step function in the vicinity of free surface, and the traditional convection schemes designed for convection-diffusion equations perform poorly in transporting the step-function. It has been found that convection schemes with some degree of downwind bias resolve the sharp interface much better.

Second-order central differencing (CD) scheme is often used in large eddy simulation (LES) and direct numerical simulation (DNS) in favor of its low-dissipation that is critical to accurately resolve turbulent structures. However, CD scheme is inherently unstable, giving troubles for cases involving fine meshes and small effective viscosity (large cell Reynolds number). One should consider using a stabilized form of central differencing.

3. COMPUTATION

At runtime, a few decisions need to be made:

a) In modern computers, choose the number of processors so that you use 50,000 to 200,000 grid points per processor.

b) To maximize performance, try to distribute the load evenly between nodes. For instance, if running in a Linux cluster with 2 dual-core processors (4 cores) per node, and your case needs 6 cores, you can distribute your load in two nodes using 4-2 or 3-3 configurations. The second balances the load per node better.

c) Modern workstations with shared memory are available with up to 48 processors, though much larger specialized systems are produced. High-performance clusters are typically cheaper per processor for large systems (thousands of cores) but use distributed memory. Shared-memory systems allow all processors access all memory, resulting in easier programming and better scalability of most applications. On the other hand, distributed memory systems provide massive number of processors for very large computations.

4. POST-PROCESSING

4.1 Visualization

A number of post processing plots should be used as a minimum sub-set of information to ensure that the correct settings have been used for each computation. This should include the following:

- Contour plots of the pressure coefficient, skin friction coefficient and y^+ of the geometry surface
- Contour plots of the boundary layer profiles along the hull geometry
- Contour and vector plots of the nominal or powered wake upstream of the plane of the propulsor (Care should be taken to ensure that this plane does not lie within the propulsion disc/volume)

Reasonable checks should be carried out to ensure that these plots are smooth and continuous. In particular, regions of specially bad grid quality should be evaluated to check if the solver can handle properly less than optimal grids without causing unphysical artifacts.

4.2 Verification and Validation

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

5. USEFUL WEBSITES AND REFERENCES

1. MARNET-CFD Best Practice Guidelines for Marine Applications of CFD (https://pronet.wsatkins.co.uk/marnet/guidelines/guide.html)
2. ERCOFTAC Best Practice Guidelines. (http://www.ercoftac.org/index.php?id=77)
3. http://www.cfd-online.com/Wiki/Best_practise_guidelines
4. http://www.cfd-online.com/Links/onlinedocs.html#bestpractice
5. http://www.nafems.org/resources/cfd_guidance/
6. ITTC, 1999, "Uncertainty analysis in CFD, Verification and Validation Methodology and Procedures", No. 7.5-03-01-01.
7. ITTC, 1999, "Uncertainty analysis in CFD, Guidelines for RANS codes", No. 7.5-03-01-02.
8. ITTC, 1999, "CFD User's Guide", No. 7.5-03-01-03.
9. ITTC, 1999, "CFD Verification", No. 7.5-03-01-04.
10. Eca L. Vaz G. and Hoekstra M., 2010. "Code Verification, Solution Verification and Validation In RANS Solvers", Proc. of ASME 2010 29th Intl Conference on Ocean, Offshore and Arctic Engineering, China.
11. Tao X. and Stern F., 2010, "Factors of Safety for Richardson Extrapolation", Journal of Fluids Engineering, Vol. 132, 061403.
12. ASME Committee PTC-61, 2009, ANSI Standard V&V 20. ASME Guide on Verification and Validation in Computational Fluid Dynamics and Heat Transfer.
13. Eca L. and Hoekstra M., 2008: "Testing Uncertainty Estimation and Validation Procedures in the Flow around a Backward Facing Step", 3rd Workshop on CFD Uncertainty Analysis, Lisbon.

14. Eca Luis and Hoekstra Martin, 2006: "Discretization Uncertainty Estimation based on Least Squares version of the Grid Convergence Index", 2nd Workshop on CFD Uncertainty Analysis, Lisbon.
15. Eca L. and Hoekstra M., 2002: "An Evaluation of Verification Procedures for CFD Applications", Proceedings of the 24th Symposium on Naval Hydrodynamics, Japan.
16. Stern F. et al. 2001, "Comprehensive Approach to Verification and Validation of CFD Simulations Part 1: Methodology and Procedures", Journal of Fluids Engineering, Vol. 123, December.
17. AIAA (1988), "AIAA guide for the verification and validation of computational fluid dynamics simulations", AIAA G-077-1998.
18. Rizzi, A. & Voss, J. (1998), "Towards establishing credibility in computational fluid dynamics simulations", AIAA Journal, vol. 36, no. 5, pp. 668-675.
19. Roache, P.J. (1998), "Verification and validation in computational science and engineering", Hermosa Publishers, Alberquerque.

6. EXAMPLE FROM G2010 WORKSHOP

An example of the application of these guidelines is illustrated by one of the Gothenburg 2010 (G2010) workshop test cases. The example chosen was Test Case 2.1, the KCS hull form without rudder and calm water conditions with zero sinkage and trim. The Reynolds number was defined as Re = $1.4\text{x}10^7$ and the Froude number defined as Fr = 0.26. The length of the hull Lpp was defined as 230.0 m at full scale with a scale ratio of λ = 31.6 for the model scale measurements.

The Froude number defines the free stream speed of

$U = Fr\sqrt{g(Lpp/\lambda)} = 0.26*\sqrt{(9.81*230/31.6)}$
$= 2.196$ m/s.

The wavelength, λ_W corresponding to this Froude number for the model scale geometry is given by:

$\lambda_W = 2\pi(L_{pp}/\lambda)\,Fr^2 = 2\pi*(230/31.6)*0.26^2$
$= 3.0915$ m.

The Reynolds number defines

$C_F = 0.075/(\log_{10}Re-2)^2$
$= 0.075/(\log_{10}(1.4\text{x}10^7)-2)^2 = 0.002832$

and for $y^+=1$ the first cell height

$y = y^+(L_{pp}/\lambda)/(Re\sqrt{(Cf/2)}$
$= (230/31.6)/(1.4\text{x}10^7\sqrt{(0.002832/2)})$
$= 1.3816\text{x}10^{-5}$ m.

Other first cell heights simply scale the distance for $y^+=1$ so $y^+=30$ is

$y=30*1.3816\text{x}10^{-5}$ m.

Checks of the geometry file provided as an IGES file showed that the geometry was built with a suitable tolerance.

The domain sizes chosen for the majority of the flow calculations carried out for the G2010 Test Cases 2.1 workshop had an upstream boundary of approximately 1 L_{pp} from the bow, a downstream boundary of approximately 2 L_{pp} from the stern, a side boundary 1 L_{pp} from the plane of symmetry and a bottom boundary of 1 L_{pp} from the keel. 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 0.5-1 L_{pp} from the keel and other contributors defined the top boundary at the deck at 0.025 L_{pp}. The choice of top boundary position was dependent on details of the chosen boundary condition.

Another variant on the domain size was to choose to match the width and depth of the domain to the width and depth of the towing tank in which the model scale hull was measured. This is appropriate for validation cases where detailed comparison with measurements is required.

The majority of contributions to the G2010 workshop Test case 2.1 used hexahedral cells with expansion ratios between 1.2 and 1.5 in the boundary layer. A number of techniques were used to create the hexahedral cells, some used single block methods, some multi-block methods and other overset and Cartesian cell methods so most of the techniques that are described in these guidelines were used. In addition, some contributors used unstructured prism 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 3.0915/40 = 0.07728 m. All contributors used at least this number of points in the axial direction on the ship hull except where coarse grids were defined for grid resolution studies.

Most contributors used a VoF method to define the free surface wave pattern.

Nearly all contributors used a variant of the k-ω turbulence model with some contributors using Reynolds stress models in addition to the two equation model. Some contributors used wall functions, with some modifications to account for pressure gradients but most used near wall boundary conditions with $y^+=1$.

Second order accurate numerical schemes were used for this test case with some contributors using higher order methods. Most contributors used a special scheme for the VoF to improve the free surface capturing.

A range of different computational resources were used for these cases, from workstations to large scale supercomputers, but the majority of contributors used some form of parallel processing techniques to reduce the elapsed time for the computations.

Convergence criteria of at least three orders of magnitude reduction in the residuals were used by all contributors.

Contour plots of the wave patterns and longitudinal plots of the wave profile at a number of positions were produced by every contributor and compared with the measured data.