
 INTERNATIONAL TOWING TANK CONFERENCE	ITTC – Recommended Procedures and Guidelines	7.5-03 04 - 01 Page 1 of 16	
	Guideline on Use of RANS Tools for Manoeuvring Prediction	Effective Date 2011	Revision 00

Table of Contents

<p>Use of RANS Tools for Manoeuvring Prediction2</p> <p>1. PURPOSE OF GUIDELINE.....2</p> <p>2. SIMULATION APPROACH.....2</p> <p>2.1 General Considerations 2</p> <p>2.1.1 Scale 2</p> <p>2.1.2 Governing Equations of the Fluid 2</p> <p>2.1.3 Turbulence Model 3</p> <p>2.1.4 Propulsion Model 3</p> <p>2.1.5 Computational Grid 4</p> <p>2.1.6 Coordinate Frame 5</p> <p>2.1.7 Boundary Conditions 5</p> <p>2.1.8 Free surface treatment 6</p>	<p>2.2 Direct Manoeuvring Simulation 7</p> <p>2.2.1 Motion equations of the ship 7</p> <p>2.2.1.1 Coupling of ship motions & flow 8</p> <p>2.3 Simulation of Forced Motions 8</p> <p>2.3.1 Forced ship motions 9</p> <p>2.3.2 Analysis of predicted forces 9</p> <p>3. PRELIMINARY STEPS.....9</p> <p>4. EXAMPLES10</p> <p>4.1 Direct Manoeuvring Simulation 10</p> <p>4.2 Simulation Based on Derivatives 10</p> <p>5. REFERENCES14</p>
---	---

Prepared by	Approved
26 th ITTC Manoeuvring Committee	26 th ITTC
Date 01/2011	Date 09/2011

	ITTC – Recommended Procedures and Guidelines	7.5-03 04 - 01 Page 2 of 16	
	Guideline on Use of RANS Tools for Manoeuvring Prediction	Effective Date 2011	Revision 00

Use of RANS Tools for Manoeuvring Prediction

1. PURPOSE OF GUIDELINE

RANS tools, i.e. numerical methods for solving the Reynolds Averaged Navier Stokes equations for viscous turbulent flows, can be applied to predict the manoeuvring behaviour of a vessel. This is achieved either in a direct way, using the considered RANS code for predicting the trajectory and, more in general, the 6 Degrees of Freedom (DoF) motion due to the movement of an appendage such as the rudder, or using it to calculate the hydrodynamic forces and moments acting on the ship or ship model during forced motions. The latter results can be used to determine manoeuvring derivatives for manoeuvring predictions.

A description of different techniques is presented from the practical point of view, together with recommended practices to obtain feasible manoeuvring prediction results. The numerical techniques used to discretise the involved partial differential equations, e.g. finite difference method or finite volume method, to model the turbulence of the flow and to generate grids have been described in many publications (e.g. Anderson et al. 1984, Blazek 2001, Ferziger and Peric 2002, Hirsh 1988, Wilcox 1993).

The present guideline is dedicated to surface ships in unrestricted waters, where usually only four degrees of freedom (surge, sway, yaw, roll) are relevant for manoeuvring.

2. SIMULATION APPROACH


2.1 General Considerations

2.1.1 Scale

In principle RANS simulations can be done for the full scale ship, avoiding any scale effect. In practice however, most simulations are performed for the model rather than the full scale ship because computations for Reynolds numbers of the order 10^9 are not fully validated yet and yield much more numerical difficulties than for Reynolds number at model scale, being 2 orders of magnitude smaller. In addition, prediction results for the model can be judged as a whole by comparing them with the results of a few selected free model tests. This “hybrid” prediction procedure seems especially attractive for towing tanks.

2.1.2 Governing Equations of the Fluid

The Navier-Stokes (NS) equations and the continuity equation describe the conservation of momentum and mass in a viscous turbulent incompressible flow and are best suitable to describe the flow around a ship. In order to work with mean values of all flow variables (e.g. velocities, pressure) instead of instantaneous values, the RANS equations are obtained by averaging the NS equations. This averaging can be seen as time averaging in case of a steady mean flow, but has to be understood as ensemble averaging in case of an unsteady

	ITTC – Recommended Procedures and Guidelines	7.5-03 04 - 01 Page 3 of 16	
	Guideline on Use of RANS Tools for Manoeuvring Prediction	Effective Date 2011	Revision 00

mean flow (e.g. Wilcox 1993, Cebeci et al. 2005). As a result of the averaging, the RANS equations contain some new unknown terms representing the effect of the turbulence on the flow. In order to solve the set of conservation equations, these terms are approximated by a turbulence model. The reason for doing so is that if not, the required space and time resolution for solving the NS equations directly would be impracticable (probably still in the next decades) for a turbulent ship flow.

2.1.3 Turbulence Model

Any turbulence model used by usual RANS applications can also be used for manoeuvring tasks. The most popular models are the $k-\epsilon$ and $k-\omega$ models (Launder and Spalding 1974, Wilcox 1993) and several variants with and without using wall functions, which allow a significant more coarse resolution of near wall regions. When looking for accurate prediction of complex flow phenomena however, e.g. detailed flow separation, more sophisticated turbulence models like Reynolds Stress Models (RSM) could be a better choice (Launder (1975)). However, such models are less validated and often less robust and more time consuming than the classical two-equations models mentioned above.

Results presented at the last CFD Workshop held in Gothenburg (2010) have shown a strong dependency for both the resistance and the velocity field on the turbulent model, however, the experience from published results and workshops shows that the dependence of side force and yaw moment, i.e. of those forces which are most significant for manoeuvring, is less significant than could be expected. The reason is that these hydrodynamic forces are certainly viscosity dependent but firstly domi-


nated by pressure. In fact, satisfactory results can be achieved even using wall functions as they do not deteriorate the quality of the predictions to the same extent than when predicting resistance.

2.1.4 Propulsion Model

Disregarding cases where RANS tools are used for predicting forces on the bare hull only, e.g. to determine coefficients for hull forces in a modular mathematical model, the appendages have to be taken into account for manoeuvring tasks. Inclusion of rudders and even bilge keels has become usual in RANS applications. This complicates the grid generation and probably also some flow aspect which can lead to increased convergence difficulties, but does not really represent a problem.

The main issue is how to treat the propeller(s), which is crucial for simulating the rudder inflow correctly when rudders are placed behind propellers. Taking the real geometry of the propeller into account and considering the rotating propeller during the RANS simulation is possible (Carrica et al. 2008b) but extremely time consuming. Thus, body forces, which are added to the right hand sides of the RANS equations, are frequently used to approximate the effect of the propeller on the flow. These forces are distributed over the grid region corresponding to the spatial position of the propeller and are calculated so that they yield the propeller thrust and moment.

Body force models, mostly based on potential flow codes like vortex-lattice or panel methods, are often used for approximating the propeller effect including slip stream and swirl, which may also influence aspects of the flow like rudder stall angle, risk of cavitation, etc.

 INTERNATIONAL TOWING TANK CONFERENCE	ITTC – Recommended Procedures and Guidelines	7.5-03 04 - 01 Page 4 of 16	
	Guideline on Use of RANS Tools for Manoeuvring Prediction	Effective Date 2011	Revision 00

The body force distribution inside the propeller region may be calculated in every new time step or in some larger time intervals, based on the current propeller inflow obtained during the RANS simulation and on the propeller rpm. This can be done either interactively, running the used potential code each time again, or determining the forces in grid cells within the propeller region from a data base calculated beforehand for the considered propeller. Fig.1 shows the cylindrical body force region (rectangle) and the effect of the body forces on the axial velocity in the longitudinal central plane.

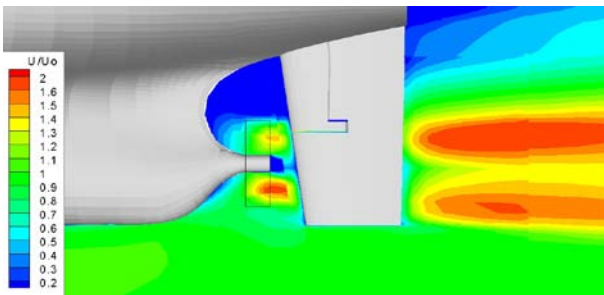


Fig.1 Body force region and effect on the flow

The choice of the propulsion point, corresponding to the full scale or to the model scale, should be decided following similar criteria as for model tests (see procedure 7.5-02-06-02). A way for determining the correct propeller rpm before starting the manoeuvre simulation is to calculate the flow for the steady straight ahead motion of the ship at the given approach speed for different rpm's and to determine the one which makes the total longitudinal force equal to the desired value (e.g. zero or estimated frictional deduction).

A proper strategy for the propeller rpm during the manoeuvre, resembling the real behaviour in full scale where the rpm often varies depending on torque, can also be implemented.

2.1.5 Computational Grid

Commercial grid generators are widespread, but also open source software is getting more popular recently. Block-structured grids, often including non-matching interfaces, and unstructured grids with several millions cells have become usual for manoeuvring applications.

Contrary to many CFD applications for ship resistance or propulsion, the nature of the problem now requires a grid covering the surroundings at both sides of the ship.

Not only for turning the propeller but also to deflect the rudder within direct manoeuvring simulations, a RANS code with sliding grid or overlapping grid capability is needed (Carrica et al. 2008a, Muscari 2008 and Durante 2010). In the later case a considerable amount of computational effort is required for transferring flow information from one grid part to the other. Otherwise and whenever possible, the grid is kept unchanged during the computation in order to not deteriorate its quality which directly influences the convergence behaviour and the quality of the results. However, this is obviously not possible in many cases of interest for example when considering squat in shallow water or approaching a quay. In such cases a suitable grid deformation technique can be an alternative to overlapping grids (Ji 2010).

The grid can be generated in several ways and many different grid topologies can be chosen. The outer boundaries of the grid mostly consist in planes delimiting a box (hexahedron) surrounding the ship. Fig.2 shows a typical configuration for a manoeuvring application for a double body in deep water.

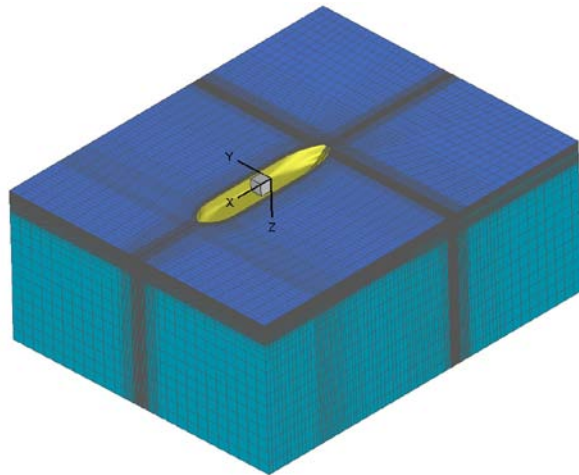


Fig.2 Grid and boundaries of hexahedral computational domain

The grid has to cover the interesting flow domain in such a way that non-physical boundaries (see 2.1.7) are far away of the region of interest, i.e. the ship vicinity. Typical dimensions of a grid are 3-5 ship lengths in longitudinal direction, 2-3 in transverse direction and one length in vertical direction for deep water.

The near wall region has to be meshed so that the requirements of the used turbulence model are fulfilled (e.g. Wilcox 1993, Menter 1994). In any case, a certain number of grid points (say 20) within the boundary layer have to be placed. For the reasons mentioned in subsection 2.1.3 regarding the influence of viscosity on side force and yaw moment, wall functions are often used for manoeuvring cases.

2.1.6 Coordinate Frame

If the flow computation is made in a ship fixed coordinate frame, i.e. if the conservation of momentum is stated in terms of its components in a ship fixed coordinate system, inertial body forces, e.g. centrifugal and Coriolis forces, have to be added to the RANS equa-

tions. These forces are usually treated explicitly during the computation and could affect the stability and convergence of the computation if they are considerably larger than the hydrodynamic forces themselves.

On the other hand, if the flow computation is made in an earth fixed or inertial coordinate frame, no inertial forces have to be added but cell boundary velocities will have to be considered in order to calculate the correct mass and momentum fluxes through the cell sides; see for instance Ferziger and Peric 2002. Both procedures are mathematically equivalent. The numerical advantages of one or the other procedure seem not significant for typical manoeuvring applications.

2.1.7 Boundary Conditions

The boundary conditions (BC) are crucial for the accuracy of the numerical solution. Setting non-physical boundary conditions such as undisturbed flow (Dirichlet) or zero-gradient (Neumann) too close to the ship will affect the results. The way BC are imposed within the numerical technique may change from code to code but does not differ for manoeuvring tasks from other applications. However, during manoeuvring simulations there are often no longer unambiguous inlet or outlet borders of the computational domain but mixed forms.

In unsteady flow cases, the BC may have to be updated in the course of the simulation according to the instantaneous ship motion.

At an “inlet” border for instance, far in front of the ship (e.g. $1 L_{pp}$) the absolute velocity is zero (in absence of current and waves).

Within a ship fixed frame however, inlet velocities are relative velocities and therefore of equal magnitude but opposite sign than the velocity resulting in the considered point of the boundary from the translation and rotation of the ship fixed coordinate system:

$$u_{\text{inlet}} = - (u - y r)$$

$$v_{\text{inlet}} = - (v + x r)$$

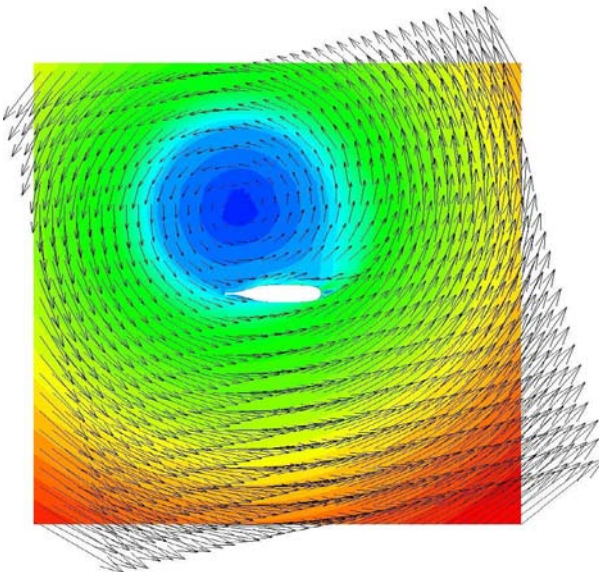


Fig.3 Velocities in horizontal plane around a ship in steady turning to starboard with drift angle 22°

A pressure BC, either zero pressure for double body flow or undisturbed hydrostatic pressure distribution for free surface flow, has proven to be advantageous for the “outlet” border far behind the ship (e.g. 2-4 L_{pp}).

At the sides of the computational domain, e.g. placed 1-2 L_{pp} away from the ship, the velocities may also be given, but these borders could also be treated as inlet and outlet boundaries, for instance in case of a steady oblique towing motion at large drift angle.

At rigid walls like the hull, a “no slip” BC is mostly set, ensuring that the fluid particles have the same velocity as the wall. Sometimes however, it is convenient to consider a wall without any friction, a “free slip” wall, for instance to delimit the computational domain. Note that, if planar, such walls behave similar to symmetry planes.

The bottom of the computational domain can be seen as a free slip wall placed far below the ship for deep water (e.g. one L_{pp}). Same can be chosen for the top border of the considered hexahedral domain, placed at the waterline in case of double body flow or at some distance (e.g. 0,1 – 0,3 L_{pp}) above the waterline in case of a free surface flow.

Note that during manoeuvres often no real inlet and outlet boundaries exist and a border of the computational domain may change its character during the simulation. For these reasons some adapted “mixed” BC taking this feature into account have proven to be very advantageous. Hereby the velocities are given if the flux is directed into the domain only and they are let free otherwise. This has been done at the left, upper and lower lateral borders in the example of Fig.3, while undisturbed pressure was assumed at the right border. The calculated velocity field differs from the undisturbed field in the close vicinity of the ship only.

2.1.8 Free surface treatment

Computations can be performed taking the water free surface into account or not. The latter approach is reasonable for a slow ship in deep water and requires significantly less computational effort (e.g. factor 10). However, even at low Froude numbers, the underwater shape and thus the forces could change signifi-

cantly if the sinkage and trim of the vessel vary at large drift angle or yaw rate. A way to take such changes into account would be including the free surface and using a 6 DoF motion model (see below) letting the ship free to sink and trim during the simulation.

Including the water free surface however, even having become more standard in the last years, leads not only to more computational time but also to increased numerical difficulties. In particular, reflection of the waves generated by the ship on non-physical or open boundaries (outlet) should be avoided. Among other techniques to avoid such reflections, a strong coarsening of the grid towards the outlet has proven to be very efficient in damping the outgoing waves preventing reflections in a rather rude manner. This procedure however would not be applicable if the considered boundary changes its type (e.g. from outlet to inlet) in the course of the simulated manoeuvre.

2.2 Direct Manoeuvring Simulation

Rudder manoeuvres like zigzag tests and turning circle tests are simulated by solving together the motion equations of the ship, considered as a rigid body, and the RANS equations for the fluid. The rudder(s) is (are) turned according to the desired manoeuvre during the simulation. This kind of manoeuvring simulation is extremely time-consuming but, since there is no mathematical model for the hydrodynamic forces involved, in principle easier than by means of manoeuvring derivatives. It will represent the best approach once comprehensively validated.

2.2.1 Motion equations of the ship

In order to predict the manoeuvre, the rigid motion equations of the ship in 3-DoF, 4-DoF or even in 6-DoF are numerically integrated in time with a proper discretisation scheme, e.g. Euler implicit, Runge-Kutta, etc. In most applications, provided large accelerations are not expected, the Euler explicit scheme can be used as well. The considered motion parameters should be properly defined by means of an earth-fixed or “inertial” coordinate system, a ship-fixed coordinate system and/or with help of an intermediate or “hybrid” coordinate system to uniquely define angles and translations. The singularity (gimbal lock, typically for $\cos\theta=0$) occurring when using Euler angles is not relevant for a surface ship.

An example of motion equations in four degrees of freedom (4 DoF) for a free sailing (rigid) ship or model, written in a hybrid coordinate system which follows the ship motions excepting roll, reads:

$$m \left[\dot{u} - \dot{\psi} v - x_G^* \dot{\psi}^2 + z_G^* (2 \dot{\psi} \dot{\phi} \cos \phi + \ddot{\psi} \sin \phi) \right] = X$$

$$m \left[\dot{v} + \dot{\psi} u + x_G^* \ddot{\psi} + z_G^* \left((\dot{\psi}^2 + \dot{\phi}^2) \sin \phi - \ddot{\phi} \cos \phi \right) \right] = Y$$

$$\begin{aligned} & \left(I_{yy} \sin^2 \phi + I_{zz} \cos^2 \phi \right) \ddot{\psi} + 2 \left(I_{yy} - I_{zz} \right) \dot{\psi} \dot{\phi} \sin \phi \cos \phi \\ & - I_{xz} \left(\ddot{\phi} \cos \phi - \dot{\phi}^2 \sin \phi \right) + m x_G^* \left(\dot{v} + u \dot{\psi} \right) \\ & + m z_G^* \sin \phi \left(\dot{u} - v \dot{\psi} \right) = N \end{aligned}$$

$$\begin{aligned} & I_{xx} \ddot{\phi} - I_{xz} \ddot{\psi} \cos \phi + \left(I_{zz} - I_{yy} \right) \dot{\psi}^2 \sin \phi \cos \phi \\ & - m z_G^* \cos \phi \left(\dot{v} + u \dot{\psi} \right) = K \end{aligned}$$

The surge and sway velocities u and v are the components of the velocity of the chosen ship origin O in the horizontal longitudinal and transversal directions x and y of the hybrid

coordinate system, respectively. The Euler angles φ and ψ are the rotations around the x- and y-axes respectively and describe the roll and yaw motions of the ship. The dots in the above equations denote time derivatives. m is the mass of the ship or model and x_G^* and z_G^* are the coordinates of the center of gravity G in the ship fixed system. It is assumed that $y_G^* = 0$.

I_{xx}, I_{yy}, I_{zz} are the moments of inertia about the ship fixed axes through the origin O and I_{xz} is the product of inertia. It is assumed that $I_{xy} = 0$ and $I_{yz} = 0$. X and Y (longitudinal and side force) are the components in the hybrid system of the external force acting on the ship. K and N (roll and yaw moment) are the components in the hybrid system of the moment of the external forces.

Since heave and pitch motions are neglected, the state of movement of the ship is defined by the position of O (earth fixed coordinates), its velocity vector $(u, v, 0)$, the Euler angles φ, ψ and the angular velocity vector $(\dot{\varphi}, 0, \dot{\psi})$. The time history of these variables can be obtained by integrating the motion equations in time numerically. For this purpose, the hydrodynamic forces and moments on their right hand sides are needed.

The hydrodynamic forces and moments appearing in the right hand side of the motion equations are calculated in the course of the time integration by simulating the flow at every new time step. Note that even if heave and pitch are not relevant for manoeuvring prediction in most cases, the ship/model should be free to sink and trim during the RANS simulation in order to get the hydrodynamic forces for the most realistic floating condition as possible.

This is easily fulfilled when making simulations with a fully 6-DoF motion model.

Note that it is possible to disable selected motions during the simulations and also to add some external forces, like a frictional deduction force resembling the free model test condition.


2.2.1.1 Coupling of ship motions & flow

The coupling between the ship motions and the flow is crucial for determining the hydrodynamic forces. If only moderate ship accelerations are involved (as usual during manoeuvres) this coupling can easily be implemented in an explicit manner: In every new time step of the simulation the RANS code is used to calculate the forces acting on the ship. Subsequently the motion equations yield the motion parameters for the next time step. Finally, the boundary conditions and inertial forces (if present) are updated before starting a new time step.

More sophisticated and in general more robust techniques have been recommended, however, at the cost of (significantly) larger computational effort.

2.3 Simulation of Forced Motions

Due to the enormous computational effort required for the direct simulation of manoeuvres described above, another strategy has gained popularity instead. It consists in simulating the usual PMM or CPMC tests numerically, solving the RANS equations around the ship or ship model when performing prescribed motions. Compared to direct manoeuvring simulations, this prediction procedure has the same advantages and disadvantages than between free and captive model tests. From the

	ITTC – Recommended Procedures and Guidelines	7.5-03 04 - 01 Page 9 of 16	
	Guideline on Use of RANS Tools for Manoeuvring Prediction	Effective Date 2011	Revision 00

computational point of view however, it is definitively more robust and less time consuming.

The strategy fully resembles the classical, well accepted PMM tests followed by the determination of derivatives and seems already practicable for commercial applications. Nevertheless a mathematical model (e.g. a set of coefficients of Abkowitz type or coefficients of formulae for diverse forces of a modular simulation method) is involved, introducing a further source of uncertainty into the prediction.

2.3.1 Forced ship motions

Motion equations are not solved in this case. Selected motions, e.g. harmonic pure sway, pure yaw, etc. are imposed. There are different ways for imposing the motions. In order to resemble CPMC tests or to reproduce measured motions during free model tests it can be advantageous to read a file containing the time histories for the motion parameters.

Note that also in this case and disregarding that the ship motion is given, it would be best to let the ship or model free to sink and trim during the RANS simulation. However, contrary to subsection 2.2.1 where the motions are predicted anyway and just 2 more DoF should be considered for including sinkage and trim, this is less straightforward now and leads to a combination of given and predicted motions.

2.3.2 Analysis of predicted forces

The analysis of the predicted time histories of the longitudinal and transverse forces X , Y and the roll and yaw moments K , N is the same as when performing PMM or CPMC model tests. Moreover, since no artificial time lag between predicted forces and prescribed


motions arise and no inertial forces have to be subtracted (no filters, no swinging masses), the analysis is easier than performing model tests.

Similar to when performing model tests there are different ways of determining the manoeuvring derivatives and the “virtual” test program has to be decided according to this and to the used mathematical model (e.g. the derivatives to be determined).

3. PRELIMINARY STEPS

The first step of any numerical investigation for manoeuvring consists in analysing the considered case and taking decisions like limiting the calculations to double body flow or taking the free water surface into account, considering the free sinkage and trim or not, performing the simulations for the ship model or for the full scale ship. This is followed by the proper choice of a turbulence model, discretisation schemes, grid and time resolution, and the choice of the boundary conditions at the borders of the grid.

In addition, several parameters of the used code have usually to be chosen as well, for instance: the number of (outer) iterations within each time step, the number of (inner) iterations within an outer iteration, values for diverse under-relaxation factors, among others. Depending on the code, other settings could also be required and have a strong influence on the result of the computations. For these reasons, experience in viscous flow computations and insight about the RANS code going to be used are prerequisites for successful CFD based manoeuvring prediction.

 INTERNATIONAL TOWING TANK CONFERENCE	ITTC – Recommended Procedures and Guidelines	7.5-03 04 - 01	
	Guideline on Use of RANS Tools for Manoeuvring Prediction	Page 10 of 16 Effective Date 2011	Revision 00

4. EXAMPLES

4.1 Direct Manoeuvring Simulation

Due to the high requirements on the used code as well as to the level of expertise and large computational capability required, this method is not yet well established in practice and will not be treated here. An example by Carrica et al. 2008 can be seen in the proceedings of the workshop SIMMAN'08.

4.2 Simulation Based on Derivatives

The technique outlined above is applied here to predict the manoeuvrability of a Very Large Crude Carrier (VLCC), namely the tanker KVLCC1, used as a benchmark test in SIMMAN'08. Due to the low Froude number of the considered tanker and because negligible heel angles are expected during its manoeuvres all RANS simulations are performed without taking the water free surface into account.

Table 1 Main particulars of KVLCC1

L_{pp}	320.0 m
B	58.0 m
T	20.8 m
∇	312738 m ³
C_B	0.8101
L_{CB}	3.48 %
GM	5.71 m
i_{xx}/B	0.375
i_{zz}/L_{pp}	0.25
Rudder lateral area	136.7 m ²
Rudder helm rate	2.34 °/s
Ship speed U_0	15.5 kn

A RANS code is used to calculate the flow around the tanker at several static conditions

and during virtual pure surge, pure sway, pure yaw and combined sway-yaw tests to obtain a rather simple set of hydrodynamic coefficients of Abkowitz type, see below.

All dynamic tests are simulated using the same multi-block structured grid with about one million cells with (some) non-matching block interfaces. The semi-balanced horn rudder, embedded in an individual grid box, is not deflected during these simulations. For static cases with deflected rudder and constant drift angle and/or yaw rate only this grid box is replaced by another according to the considered rudder angle.

The grid dependency of the results has to be checked at least by means of selected calculations on different grids. In the present case the values of all forces and moments acting on the ship obtained on coarse, medium and fine grids behaved consistently and differed less than 10% from each other. Although this check cannot replace a real Uncertainty Analysis (UA) it may be a good compromise in practise.

The computations are performed on a ship fixed grid using a Cartesian non-inertial coordinate system. The standard two equations $k-\omega$ turbulence model with wall functions is used. During dynamic tests the motions are imposed through the boundary conditions and corresponding inertial forces added to the RANS equations, see Cura and Vogt (2002).

The needed CPU time for simulating dynamic tests amounts nowadays still several days per period on a single processor of a normal PC but can be much less if a parallel code is run on a cluster with hundreds of processors. The static tests usually take some few hours depending on grid resolution.

Vortex lattice data for the propeller of a typical tanker was used in the present case. The rate of revolutions was set so that the resulting thrust balanced the resistance computed during a steady straight ahead motion of the model (model self propulsion point). This rate was kept constant throughout the computations.

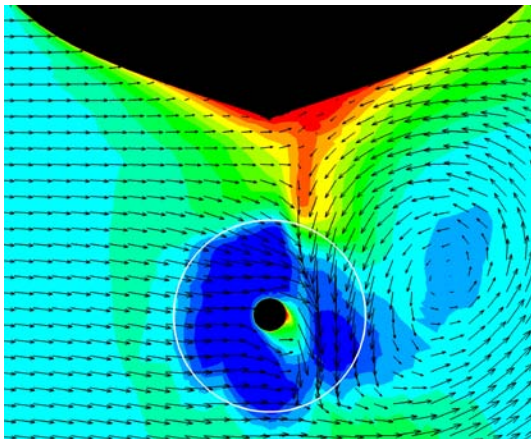


Fig.4 Snapshot of the velocity field behind the propeller during a simulated sway-yaw test

Fig.4 shows the velocity distribution just behind the propeller plane during a simulated combined sway-yaw test at a certain time when the ship is turning to starboard. The white circle indicates the body force region.

In order to obtain all manoeuvring derivatives except those depending on the rudder angle and surge velocity, five dynamic tests with large velocity amplitudes and a common non-dimensional period $T' = T U_0 / L_{pp} = 3.369$ (20 seconds in model scale) are simulated.

Similar to real tests, the non-dimensional amplitudes of the harmonic motions should be chosen so that they cover the expected range of the motion parameters during the manoeuvres.

In the present example the amplitudes were: $u' = u/U_0 = 0.10$ for pure surge, $v' = v/U_0 = 0.35$ for pure sway, $r' = rL_{pp}/U_0 = 0.70$ for pure yaw and $-0.35, 0.20$ and $-0.20, 0.40$ for two combined sway-yaw tests, respectively.

The RANS simulations were done for the tankers model (scale 1:45.7) at a speed of 1.179 m/s, instead of for the full scale. The time step chosen for the RANS simulation corresponded to 1/2500 of the motion period in all cases.

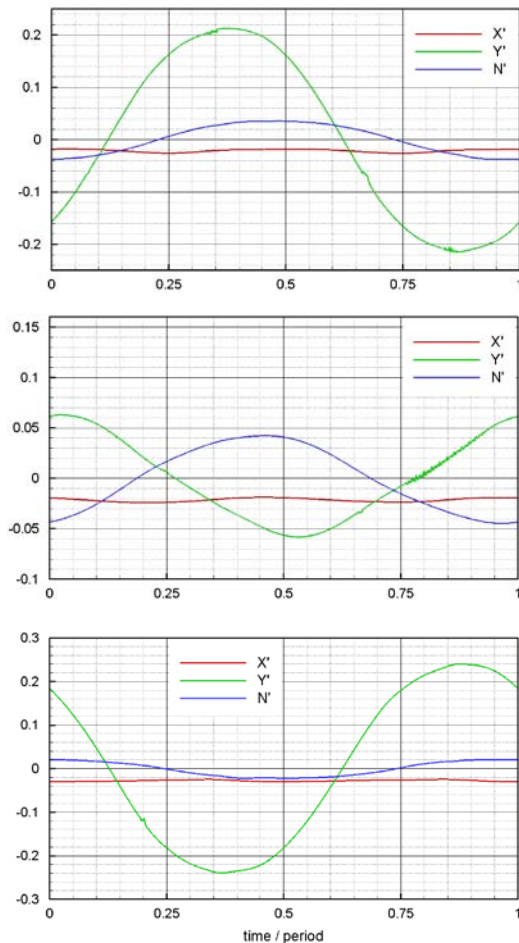


Fig.5 Forces and yaw moment during one period of a virtual pure sway and pure yaw test and a combined sway-yaw test (top to bottom)

The hydrodynamic forces and moments acting on the ship are obtained by integrating the pressure and shear stresses on the hull and appendages. The predicted time histories during simulated pure sway, pure yaw, as well as combined sway- yaw can be seen in Fig.5. The longitudinal force X' , side force Y' and yaw moment N' have been made non-dimensional with water density, ship speed, length and draught.

Rudder angle depending manoeuvring derivatives can be determined by computing rudder angle tests at several drift angles and yaw rates resulting in a total of 42 cases.

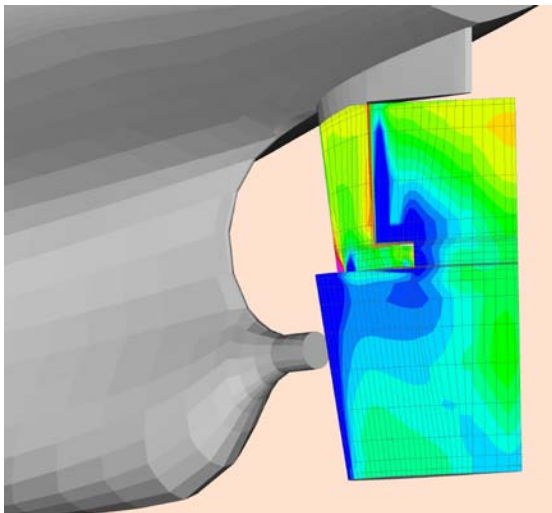


Fig.6 Stern arrangement of the virtual ship model and computed pressure on the rudder deflected 35°

Fig.6 shows the stern arrangement of the virtual model of KVLCC1 with the rudder deflected 35° to starboard. The pressure field on the rudder computed for steady straight ahead motion is influenced by the effect of the propeller, rotating to the right over the top. Negative pressure regions are depicted in blue, while positive pressure regions are in red.

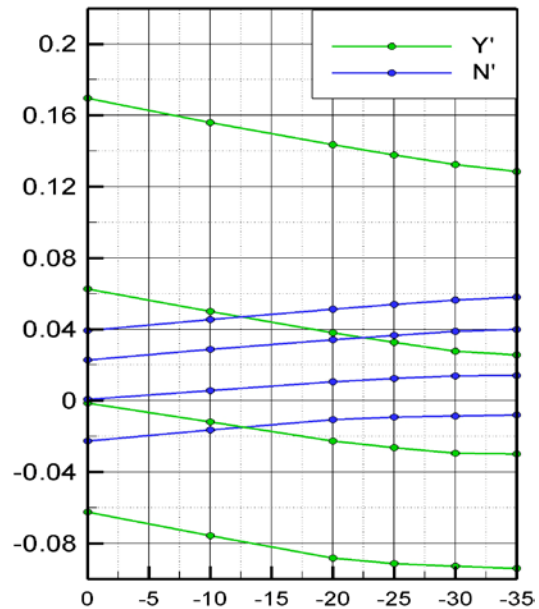


Fig.7 Computed non-dimensional side force and yaw moment during rudder angle tests at drift angle -10°, 0°, 10° and 20°

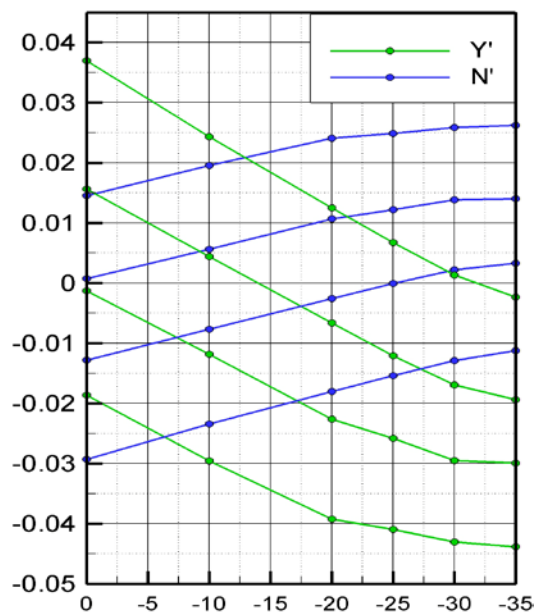


Fig.8 Computed non-dimensional side force and yaw moment during rudder angle tests at non-dimensional yaw rate -0.25, 0, 0.25 and 0.50

The computed non-dimensional side force and yaw moment acting on the hull for all static cases are summarised in Fig.7 and Fig.8 for oblique towing and steady turning conditions respectively.

The time histories of the forces obtained from the RANS simulations for the 5 dynamic tests described above are used to determine the coefficients of the mathematical model in the same way as if PMM tests would have been done. This yields the coefficients in rows 4-18 of Table 2.

Regression analysis of the data obtained from static cases with deflected rudder yields the coefficients depending on the rudder angle written in rows 1-3 and 19-23 in Table 2.

The hydrodynamic coefficients shown in Table 2 have been made non-dimensional with water density, ship speed and length and multiplied by 1000, and are used to simulate standard rudder manoeuvres according to IMO (2002). For this purpose the motion equations of the ship in four degrees of freedom (4 DOF) were used. However, the dependency of the non-dimensional magnitudes X' , Y' , N' and roll moment K' (not shown) on heel angle and roll rate was neglected since no significant roll motion was expected for the considered tanker. The sub-indices u , v , r and δ denote the surge, sway and yaw velocities and the rudder angle, respectively.

The main results of the simulated 10°/10° zigzag test starting to starboard are compared with experimental results in Fig.9 which shows the heading angle ψ and the rudder angle δ versus time. The 2nd overshoot angle predicted for KVLCC1 is slightly larger than measured and the overall agreement deteriorates with increas-

ing time. However the characteristic parameters used to judge yaw checking and initial turning ability are predicted well, Table 3.

Table 2 Manoeuvring Derivatives

0	X'_o	0	Y'_o	0	N'_o	0
1	X'_δ	0	Y'_δ	4.44	N'_δ	-2.06
2	$X'_{\delta\delta}$	-2.09	$Y'_{\delta\delta}$	-0.24	$N'_{\delta\delta}$	0.16
3	$X'_{\delta\delta\delta}$	0	$Y'_{\delta\delta\delta}$	-2.95	$N'_{\delta\delta\delta}$	1.38
4	X'_u	-2.20	Y'_u		N'_u	
5	X'_{uu}	1.50	Y'_{uu}		N'_{uu}	
6	X'_{uuu}	0	Y'_{uuu}		N'_{uuu}	
7	$X'_\dot{u}$	-1.47	$Y'_\dot{u}$		$N'_\dot{u}$	
8	X'_v	0.11	Y'_v	-24.1	N'_v	-7.94
9	X'_{vv}	2.74	Y'_{vv}	2.23	N'_{vv}	-1.15
10	X'_{vvv}	0	Y'_{vvv}	-74.7	N'_{vvv}	2.79
11	$X'_\dot{v}$		$Y'_\dot{v}$	-16.4	$N'_\dot{v}$	-0.47
12	X'_r	-0.07	Y'_r	4.24	N'_r	-3.32
13	X'_{rr}	0.58	Y'_{rr}	0.56	N'_{rr}	-0.27
14	X'_{rrr}	0	Y'_{rrr}	2.58	N'_{rrr}	-1.25
15	$X'_\dot{r}$		$Y'_\dot{r}$	-0.46	$N'_\dot{r}$	-0.75
16	X'_{vr}	13.1	Y'_{vr}		N'_{vr}	
17	X'_{vrr}		Y'_{vrr}	-40.3	N'_{vrr}	8.08
18	X'_{vvr}		Y'_{vvr}	-9.90	N'_{vvr}	-3.37
19	$X'_{u\delta}$		$Y'_{u\delta}$	-4.56	$N'_{u\delta}$	2.32
20	$X'_{v\delta\delta}$		$Y'_{v\delta\delta}$	5.15	$N'_{v\delta\delta}$	-1.17
21	$X'_{vv\delta}$		$Y'_{vv\delta}$	7.40	$N'_{vv\delta}$	-3.41
22	$X'_{r\delta\delta}$		$Y'_{r\delta\delta}$	-0.51	$N'_{r\delta\delta}$	-0.58
23	$X'_{rr\delta}$		$Y'_{rr\delta}$	-0.98	$N'_{rr\delta}$	0.43

Table 3 Characteristic parameters of 10°/10° test

10°/10°	SIM	EXP
time to attain	67 s	69 s
x_{90°	1.66 L_{pp}	1.73 L_{pp}
α_{01} [°]	8.1°	8.2°
α_{02} [°]	21.4°	19.4°
r_{max}	0.42 °/s	0.40 °/s

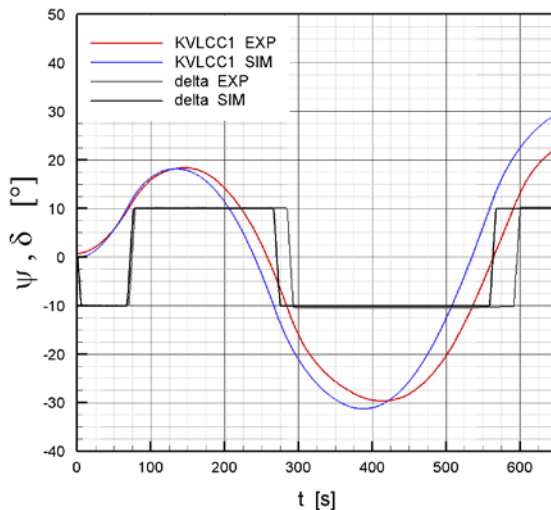


Fig.9 10°/10° zigzag test starting to starboard

Any other rudder manoeuvre of interest can be predicted as well. For instance, the result of a simulated turning circle to starboard with a rudder angle of 35° is compared with a free model test in Fig.10. The main parameters of the turning circle tests are compared in Table 4 with experiments showing good agreement. Note that the tanker fulfils the IMO recommendations with margin.

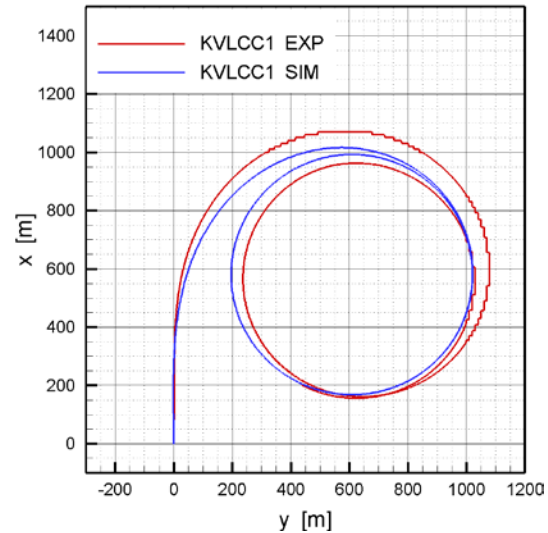


Fig.10 Turning circle test with $\delta = 35^\circ$

Table 4 Characteristic parameters of turning circle test

$\delta = -35^\circ$	SIM	EXP
x_{90° / L_{pp}	3.10	3.03
y_{180° / L_{pp}	3.13	3.25
ϕ_{st} / L_{pp}	2.58	2.44
V_{st} / V_o	0.39	0.37
r_{st} [°/s]	0.43	0.42

5. REFERENCES

Abkowitz, M.A., (1964), "Lectures on Ship Hydrodynamics - Steering and Manoeuvrability", HyA Report HY-5, Copenhagen

Anderson, D.A, Tannehill, J.C., Pletcher, R.H.

(1984), "Computational fluid mechanics and heat transfer", Hemisphere, New York

Blazek, J. (2001), "Computational Fluid Dynamics: Principles and Applications", Elsevier

Carrica, P.M., Ismail, F., Hyman, M., Bhushan, S., and Stern, F. (2008a), "Turn and zigzag manoeuvres of a surface combatant using a URANS approach with dynamic overset grids", Proceedings SIMMAN 2008, Copenhagen

Carrica, P., and Stern, F. (2008b), "DES simulations of KVLCC1 in turn and zigzag manoeuvring with moving propeller and rudder", Proceedings SIMMAN 2008, Copenhagen

Cebeci, T., Shao, J.R., Kafyeke, F., Laurendeau, E. (2005), "Computational Fluid Dynamics for Engineers", Horizons Publishing

Cura Hochbaum, A. (2006), "Virtual PMM Tests for Manoeuvring Prediction", 26th ONR Symposium on Naval Hydrodynamics, Rome

Cura Hochbaum, A., Vogt, M., Gatchell, S. (2008), "Manoeuvring prediction for two tankers based on RANS calculations", Proceedings SIMMAN 2008, Copenhagen

Durante D., Broglia R. Muscari, R., and Di Mascio A., (2010), "Numerical Simulations of a Turning Circle Manoeuvre for a Fully Appended Hull", 28th ONR Symposium on Naval Hydrodynamics, Pasadena, Los Angeles, USA.

IMO, International Maritime Organization, (2002), Resolution MSC.137(76), "Standards for Ship Manoeuvrability", London

Ferziger, J.H., Peric, M. (2002), "Computational Methods for Fluid Dynamics", Springer

Hirsch C. (1988), "Numerical computation of internal & external flows", Wiley Series in Numerical Methods in Engineering

Hirt, C.W., and Nichols, B.D. (1981), "Volume of Fluid (VOF) Method for the Dynamics of free Boundaries", J. of Comp. Physics, Vol.39


L. Ji, K. Sreenivas, D. G. Hyams, R.V. Wilson (2010), "A Parallel Universal Mesh Deformation Scheme for Hydrodynamic Applications", 28th ONR Symposium on Naval Hydrodynamics, Pasadena, Los Angeles, USA.

Launder, B.E., Reece, G.J. and Rodi, W. (1975), "Progress in the Development of a Reynolds-Stress Turbulent Closure.", Journal of Fluid Mechanics, Vol. 68(3), pp. 537-566.

Launder, B.E., and Spalding, D.B. (1974), "The numerical computation of turbulent flows", Comput. Meth. in Appl. Mech. Eng. Vol.3

Menter, F.R. (1994), "Two-Equation Eddy-Viscosity Turbulence Models for Engineering Applications", AIAA Journal, Vol.32, No.8

Muscari, R., R. Broglia and A. Di Mascio, (2008), "Trajectory Prediction of a Self Propeller Hull by Unsteady RANS Computations", 27th ONR Symposium on Naval

	ITTC – Recommended Procedures and Guidelines	7.5-03 04 - 01 Page 16 of 16	
	Guideline on Use of RANS Tools for Manoeuvring Prediction	Effective Date 2011	Revision 00

Hydrodynamics, Seoul.

Osher, S., and Sethian, J.A. (1988), "Fronts Propagating with Curvature-Dependent Speed: Algorithms Based on Hamilton-Jacobi Formulations", J. of Comp. Physics, Vol.79

Thompson, J.F., Warsi, Z.U.A., Mastin, C.W. (1985), "Numerical Grid Generation: Foundations and Applications", NorthHolland, New York

Wilcox, D.C. (1993), "Turbulence Modeling for CFD", DCW Industries, La Cañada, California