Abstract for 32nd IWWWFB, Dalian, China, 2017

Some aspects of coupling the RANS based CFD with the potential flow models for seakeeping applications

Malenica Š.[(1)], **Choi Y.M.**[(2)], **Monroy Ch.**[(1)], **Seng S.**[(1)], **Chen X.B.**[(1)] **& Vukcevic V.**[(3)]
(1) Bureau Veritas - Paris, France , (2) Ecole Centrale de Nantes, France , (3) University of Zagreb, Croatia

Introduction

Seakeeping in the present context means essentially the classical diffraction radiation problems with and without forward speed and the maneouvering. The current trends in the seakeeping simulations seem to be more and more oriented to the use of the RANS based CFD methods instead of the more classical potential flow methods. There are several reasons for that among which the inclusion of the nonlinear effects and the effects of the forward speed are probably the most important ones. Indeed, due to the very complex behavior of the free surface for large amplitude waves the potential flow methods have enormeous difficulties to model the associated nonlinear effects, especially when the wave breaking occurs. In addition, the problem of seakeeping with the forward speed also introduce the huge difficulties so that the fully consistent potential flow solution, even linear, based on the Boundary Integral Equation (BIE) technique is still missing. On the other hand, it must be recognized that the CFD methods became nowadays very efficient allowing for quite accurate evaluation of the ship seakeeping characteristics for very general operating conditions. There are however some important technical difficulties, related to the use of the CFD methods, and they are not exclusively related to the large CPU time requirements. Among these difficulties there are few of them which require particular attention. The first one is the mesh generation which might quickly become very complex and expensive. Indeed, contrary to BIE methods for potential flow where only the wetted surface of the body need to be discretized, in CFD simulations the whole fluid domain should be modelled using the 3D discretisation. This makes the proper verification of the results, in terms of stability and convergence, very complex so that quite often the proper convergence studies are missing and simply the resuts with the finest possible mesh are taken as the good ones. Another important difficulty is related to the correct propagation of the waves which are one of the most important aspect of the seakeeping. This includes both the incident waves and the perturbed waves generated by the body-fluid interactions. Within the CFD community, it looks like there is a common belief that everything can be solved by increasing the computational domain and refining the mesh. For the potential flow community this is certainly not seen as the wisest choice and the introduction of some physical aspects of the wave propagation should be made. Indeed, increasing the CFD domain and refining the mesh will not necessarily lead to the improved wave propagation model but will certainly increase the CPU time significantly. On the other hand, the potential flow methods are able to propagate the waves in a very efficient manner at least when the linear wave model is justified, which is usually true far from the body. However, the potential flow methods suffer from important numerical difficulties when the problem becomes highly nonlinear, which usually happens close to the body. All this suggest that combining the RANS and the potential flow methods might be the best approach to exploit the advantages of both methods. The main purpose of this paper is to discuss those hybrid methods. Lot of work has been already done on the subject but it looks like there is still large room for significant improvements. Within the context of seakeeping simulations, the questions we should ask ourselves are: Will the far field behavior of the wave system influence the local pressure/stress distribution at the body? Is it enough to artificially damp the wave system using some numerical tricks or it is necessary to properly simulate the waves in the whole fluid domain? What is the optimal size of the CFD domain? Can we reduce the CFD domain if we treat the waves more physically?

RANS based CFD numerical models

There are different numerical methods which can be used to solve the RANS equations. Any of those methods requires a limited fluid domain which is then discretized in 3D using the different numerical approaches (e.g. finite volumes, finite elements, finite differences...). Here we concentrate on the finite volume method implemented in the open-source software OpenFOAM®. In OpenFOAM®, the RANS equations are discretized in space using the arbitrary polyhedral cells. The basic unknown quantities

are the local fluid velocity $\boldsymbol{u}$, pressure p and the position of the free surface. The numerical schemes in space and time can be of different types (first, second or higher order, explicit, implicit, semi-implicit ...) but most of them end up with the iterative procedure at each time step during which the field equation together with the various boundary conditions are satisfied at a certain level of accuracy. Due to the fact that the physical domain is infinite the numerical model should stop somewhere in the fluid and the special end condition should be applied at those boundaries. In order to not apply this condition abruptly, the blending (also called relaxation) zones are usually employed (see Figure 2) in such a way that this condition is applied in a smooth manner. Physically the radiated waves travel very far from the computational domain and in principle this effect should somehow be taken into account.

Potential flow models and wave propagation

The most comprehensive way to understand the different wave systems is to assume the ideal linear potential flow in frequency domain. These simplifications do not consider some physical effects (e.g. viscosity, rotationality ...) but keeps the main aspects of the wave generation and propagation. The main unknown in the potential flow model is the velocity potential φ from which all other quantities (e.g. pressure, velocity...) can be deduced. The Fourier representation of surface waves generated by the oscillating disturbance moving at constant speed U, gives the analytical expression of the propagative wave components φ^w in the form of the single integrals over the different dispersion curves (e.g. see [2]):

$$\varphi^w = -i \sum_{D=0} \int [\text{sign}(D_f) + \text{sign}(\xi D_\alpha + \eta D_\beta)] A \exp[-i(\alpha\xi + \beta\eta)] / ||\nabla D|| dS \quad , \quad D = (f - F\alpha)^2 - k \quad (1)$$

where f is the non dimensional encounter frequency $f = \omega_e \sqrt{L/g}$, F is the Froude number $F = U/\sqrt{gL}$, k is the local wave length $k = \sqrt{\alpha^2 + \beta^2}$ and A is the amplitude function related to the singularity distribution at a given surface (body or other control sufaces) as it will be discussed later.
Depending on the nondimensional number $\tau = fF$, different dispersion curves exist as shown in Figure 1. The distance from the origin to one point at the dispersion curve defines the local wave number k and we can easily understand that the wave length ($\lambda = 2\pi/k$) can vary in between zero and the wave length related to the closest distance in between the closed dispersion curve and the origin. This fact gives rise to very complex wave systems as the ones shown in Figure 1. One of the main numerical difficulties in CFD methods is related to the correct representation of these wave systems. Let us also note that, for the case with no forward speed $F = 0$ the dispersion curve is represented by a circle around the origin and only one wave length exists $\lambda_0 = 2\pi/k_0$ with $k_0 = f^2$, giving purely circular wave system.

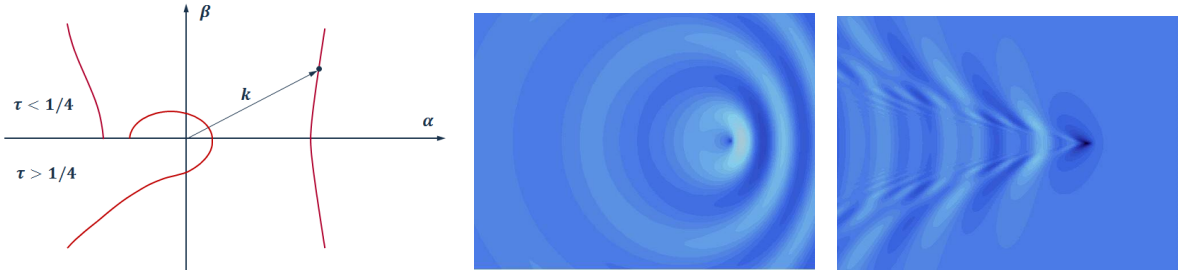

Figure 1: *Different dispersion curves (symmetric about α axis) and the wave system associated with the closed dispersion curve around origin (ring waves) and with the right open dispersion curve (V waves).*

It is important to understand that, whatever the body characteristics (shape, mass distribution...), the wave system will always be defined by the same expression (1) and only the complex ampitude A will change. This amplitude can be calculated from the distribution of the velocity potential φ and the normal velocity $\partial\varphi/\partial n$ at a given surface, using the Green's theorem.
At this stage it is important to mention that the above analysis was made for frequency domain potential flow method while we will need the potential flow solution in time domain because most of the RANS method are formulated in time domain. Clearly, the basic principles of the wave propagation remain the same in frequency and time domain and the same wave systems is generated after the transition phase. In order to take the full advantage of the potential flow model here we propose to consider the linear model based on the use of the Kelvin type time domain Green's function (the one which satisfies the linearized free surface condition) which will allow for consistent wave propagation far from the body. Within this method the velocity potential can be expressed in the following form (e.g. see [1]):

$$4\pi\Phi(\boldsymbol{x},t) = -\iint_{S_P}[\Phi(t)G_n^{(0)} - G^{(0)}\Phi_n(t)]d\boldsymbol{\xi} + \int_0^t d\tau \iint_{S_P}[\Phi(\tau)H_{\tau n}(t-\tau) - H_\tau(t-\tau)\Phi_n(\tau)]d\boldsymbol{\xi}$$
$$+\frac{U}{g}\int_0^t d\tau \int_{\Gamma_P}\{\Phi(\tau)[H_{\tau\tau}(t-\tau) - UH_{\tau\xi}(t-\tau)] - H_\tau(t-\tau)[\Phi_\tau(\tau) - U\Phi_\xi(\tau)]\}n_1 dl \quad (2)$$

where U is the forward speed in x direction, $G^{(0)}(\boldsymbol{x};\boldsymbol{\xi})$ is the impulsive part of the Green's function and $H(\boldsymbol{x};\boldsymbol{\xi},t-\tau)$ is the memory part of it. The subscripts in the above equation denotes the derivatives, and the space dependence of the different quantities in the right hand side is ommitted for clarity. Note also that, for the sake of clarity, the corresponding boundary value problem for the potential Φ was defined using the so called uniform flow approximation which is not mandatory and more complex linearizations could be employed.
What is important to understand from the above representation is that if we know $\Phi, \Phi_n, \Phi_\tau, \Phi_\xi$ at the surface S_P at each time instant (see Figure 2), we can calculate the velocity potential (and other quantities) at any point in the fluid. This fact is crucial for the coupling with the CFD code because it allows the calculation of the flow quantities in the interaction zone without solving the BIE.

Coupling

There are different important aspects of the coupling which should be considered together. In order to clarify the different methods it is useful to introduce the concepts of the domain and the solution decompositions. In Figure 2 the basic concept of the domain decomposition method is presented. In the left part of the figure, the original problem which we want to solve (floating body in waves) is presented and in the right part of the figure the problem is decomposed in different domains. The physical quantity which we are looking for is denoted by q and it can be any quantity such as pressure, velocity, wave elevation ... The Navier Stokes quantities are denoted by q^{NS} and the potential flow quantities by q^{PF}. The Navier Stokes domain is the domain represented by the quadrilateral mesh while the potential flow domain is represented is limited by the free surface and the control surface S_P.

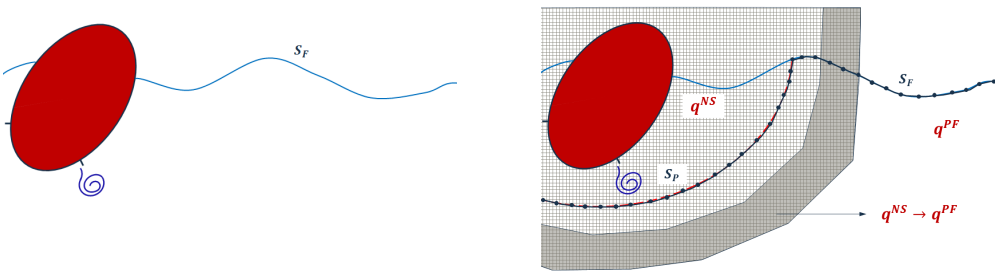

Figure 2: *Basic configuration (left - original physical problem, right - domain decomposition).*

The basic principles of the coupling are rather simple and intuitive: we calculate the different quantities using the different numerical methods in their respective domains and we make them match to each other in the interaction zone. This interaction zone, represented in gray color in Figure 2, can be any part of the fluid domain in which both solutions are defined. The choice of this interaction zone as well as the choice of the Navier Stokes domain and the potential flow domain should be made with greatest care.
In addition to the domain decomposition it is useful to decompose the total solution into two parts:

$$q^{NS} = q_0^{NS} + q_1^{NS} \quad , \quad q^{PF} = q_0^{PF} + q_1^{PF} \quad (3)$$

where q_0 denotes the known quantity and q_1 the unknown quantity. The main idea behind this decomposition is to numerically calculate only those quantities which are not known and include explicitely the quantities which are known (analytically or other). In that sense q_0^{NS} and q_0^{PF} represent the same known quantity expressed in the different way. This decomposition was first introduced in [3] , and is known under the name SWENSE (Spectral Wave Explicit Navier Stokes Equations) decomposition.
The relaxation of the RANS solution to potential flow solution is performed in the blending zone in a rather natural way by using the simple blending function $w(\boldsymbol{x})$. With the above defined solution decomposition (3) we can formally write:

$$[1-w]\mathcal{T}(q^{NS}) + w\mathcal{P}(q^{PF}) = 0 \quad (4)$$

where $w(\boldsymbol{x})$ is a smooth function equal to 0 at the begining of the blending zone and 1 at the end of the RANS domain. The operator $\mathcal{T}$ stands for the RANS operator (transport equation, continuity...) and $\mathcal{P}$ stands for the potential flow operator (BIE representation, Bernoulli equation...). It is also possible to apply the blending procedure directly on the quantities q^{NS} and q^{PF} which leads to so called explicit blending. It should also be noted that the blending function $w(\boldsymbol{x})$ could also be chosen to be time dependent in which case the blending is dynamic and this might possibly help in relaxing the RANS solutions more efficiently when and where the difference in between the solutions is larger. In any case the main goal of the blending remains the same i.e. extracting the wave part of the solution, and we should accept that this can be achieved only approximately.

Discussions

In the first attempts of the CFD methods to solve the seakeeping problem, the wavemaker was used for wave generation at one side of the domain and the total solution was damped to zero at the other side which, in fact, represents the direct simulation of the wave tank. This was probably the simplest approach to implement but certainly not the wisest one. Most popular methods nowadays [4], generate the waves through the blending zone where the solution q^{NS} is relaxed either to zero ($q_0^{NS} + q_1^{NS} \to 0$) either to incident wave $q_1^{NS} \to 0$. The main idea of the present work is to, instead of relaxing the perturbation part to zero or to incident wave, relax it to the solution given by the potential flow theory where the characteristics of the potential flow are given by the distribution of $\Phi, \Phi_n, \Phi_\tau, \Phi_\xi$ at the control surface (2).

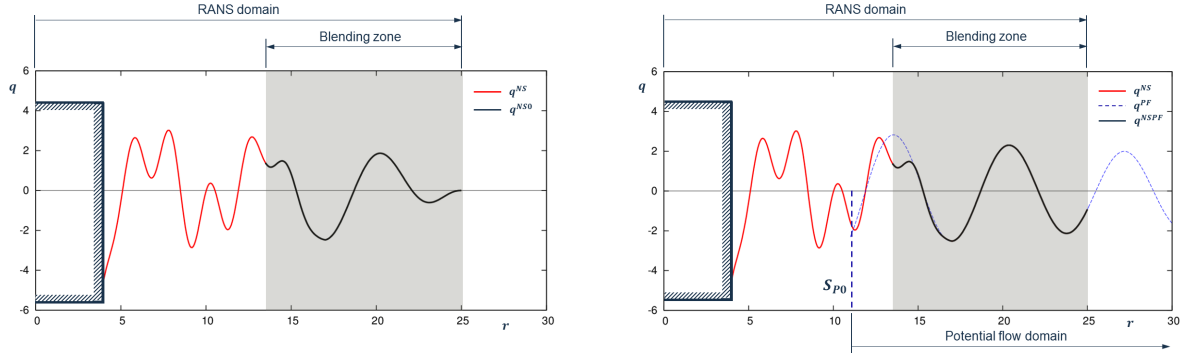

Figure 3: *Relaxation of the RANS solution to different outer solutions (left - 0 , right - potential flow).*

In Figure 3 we present two typical cases of relaxing the RANS solution toward the different outer flows. The main goal of the present studies is to quantify how the different blending methodologies affect the local solution at the body. Hopefully, relaxing the RANS solution toward the more realistic outer solution will allow for significant reduction of the CFD and blending domains size leading to important reduction of the overall CPU time. In order to be able to perform the above described coupling, several critical points need to be solved. Among them we can mention the following: deduction of the velocity potential and its derivatives from the RANS solution, separation of the potential and non-potential parts of the RANS solution, relaxation of the non-potential part, consistency of the linearization and the accurate evaluation of the potential flow, choice of the control surface for potential flow model (2) and many others. All these issues will be discussed at the Workshop in the context of the pure incident wave propagation in 2D and the nonlinear wave diffraction by vertical circular cylinder which are the problems for which the semi-analytical linear and higher order potential flow solutions are available for benchmarking purposes.

References

[1] Bingham H.B. 1994.: "Simulating ship motions in time domain.", PhD thesis, MIT

[2] Chen X.B. & Noblesse F. 1997.: "Dispersion relation and far field waves.", 12th IWWWFB, Marseille, France

[3] Ferrant P., Gentaz L. & Le Touze D. 2002.: " A new RANSE/Potential Approach for Water Wave Diffraction", Proc. Numerical Towing Tank Symposium, NuTTS2002, Pornichet, France

[4] Jacobsen, N.G.,Fuhrman & D.R.,Fredse,J. 2012. "A wavegeneration toolbox for the open-source CFD library OpenFoam", Int. J. Numer. Methods Fluids Vol. 70 (9), pp. 1073 - 1088.

[5] Vukcevic V., Jasak H. & Malenica S., 2016.: "Decomposition Model for Naval Hydrodynamic Applications, Part I: Computational Method", Ocean Engineering, Vol. 121, pp. 37-46.