RANSE Simulations of Surface Piercing Propellers

Mario Caponnetto, Rolla Research, mariocaponnetto@hotmail.com

RANSE methods have been applied to the analysis of ship propellers in open-water condition and behind ships for a good decade now. So far, these applications focused on 'conventional' propellers. These are typically modeled without consideration of the free water surface. This can be justified for conventional propellers which operate sufficiently far away from the free surface.

However, high-performance boats resort often to surface-piercing propellers which introduce several additional challenges. The propeller profiles of surface-piercing propellers have a cusped leading edge, do not end in a point (tapered forms), but have maximum thickness at the "trailing edge" which then becomes a trailing surface. Naturally the free surface changes the flow at the propeller considerably and we have to model rapid water exit and water entry. As in simulations of propellers interacting with ship hull or rudder, we have to combine turning grids with stationary grids in a RANSE simulation.

I will present results for various surface piercing propeller applications taken from the business experience of Rolla Propellers. The simulations are performed using the commercial finite-volume RANSE solver Comet.

Comet proved once again to be a rather robust code capable of handling complex free-surface applications. From a practical point of view, it is rather advantageous to have one single code to handle the hydrodynamic analyses of calm-water planing, Caponnetto (2000), seakeeping of planing hulls, Caponnetto (2002), and propulsive organs even if these are all unconventional.

Introduction

A surface piercing propellers (SPP) is a propeller working at the interface between water and air.

SPP are normally used on fast yachts for a number of reasons. When speed becomes very high (>40 knots) conventional submerged propellers present the major drawbacks of erosive cavitation and high resistance of their appendages (shafts, brackets, rudders). On a properly designed SPP erosive cavitation is normally avoided since the water vapor pocket is replaced by an air pocket; the driving shafts are completely out of the water and rudders can be avoided using steerable shafts (Arneson drives). Moreover the possibility to trim the shafts and tuning the vertical force developed by the propellers, can be very useful on planing yachts to obtain the optimum running trim.

An SPP blade, during the rotation, works in the air for about 50% of the time (developing practically no thrust), is completely submerged for 25/30% of the time and for the rest is partly submerged (in the entry and exit phases). During the completely submerged phase only the face (pressure side) should be wetted, while the back side should be surrounded by an air cavity connected to the free surface. On the back side the pressure is "nominally" equal to the atmospheric pressure, while a large pressure acts on the face.

SPP blades have an outline similar to conventional propellers, but the sections resemble those of a supercavitating propeller. The leading edge is sharp and very thin to promote cavity development; the face if highly cambered with the maximum dept close to the trailing edge (cup). The shape of the back has no hydrodynamic influence as far as its contour remains inside the air pocket. For this reason and to give the required robustness, the maximum thickness of the profile is located at the trailing edge.

Comparing to conventional propellers, there is a lack of data in the literature exploitable for the design of an SPP. The first complete methodical series tested in a cavitation tunnel have been developed by Rolla Propellers in 1991 (Rose & Kruppa, 1991). More recently an important experimental analysis, giving more insight in the physical phenomenon, has been performed by Olofsson (Olofsoon, 1996), where not only the global forces but also their instantaneous values on a single blade have been measured.

From the computational point, while in principle conventional potential flow theory could be applied to SPP design, in reality the complex free surface effects (cavity and spray), difficult to capture with a panel method, makes its accuracy not sufficient to give engineering usable data.

An attempt has been accomplished in Rolla to set up a numerical tool able to calculate SPP hydrodynamic. Due to its capability to compute complex free surfaces and having the possibility to feature sliding meshes, the RANSE solver Comet has been tested for our purpose. In this paper a brief description of the method used and some results will be presented.

1. Description of the numerical method

The flow is modeled as a two-phase flow computing both air and water flow simultaneously. The conservation equations for mass and momentum are solved in integral form using a finite volume method. The integrals are approximated using the midpoint rule. The SIMPLE algorithm couples pressures and velocities. The Reynolds stress tensor (i.e. turbulence) is modeled using the standard k-ε turbulence model. Time is discretized using an implicit Euler scheme. Demirdzic et al. (1998) give more details of the method.

At the inlet, velocity components, turbulent energy and its dissipation rate are prescribed. At the outlet, zero gradients in longitudinal direction are enforced. On the propeller surface, the no-slip condition is enforced using a wall function.

The interface between water and air is determined in a surface capturing method. We define a scalar function C ($0 \le C \le 1$) which describes the volume percentage of water in each cell. C=1 for cells filled completely with water, C=0 for cells filled completely with air. This scalar function allows us to model the two phases in our flow as one effective fluid with locally weighted material properties (viscosity μ_{eff}, ν_{eff}):

$$\mu_{eff} = C\,\mu_{water} + (1\text{-}C)\,\mu_{air}$$
$$\nu_{eff} = C\,\nu_{water} + (1\text{-}C)\,\nu_{air}$$

In addition to the RANSE and the turbulence transport equations, we solve one more convective transport equation to capture the convection of the water-air interface defined by C=0.5. The specially constructed high-resolution interface capturing (HRIC) scheme keeps the transition region from C=0 to C=1 relatively narrow and allows thus a quite sharp resolution of the water-air interface, Muzaferija and Peric (1998).

2. Description of the mesh

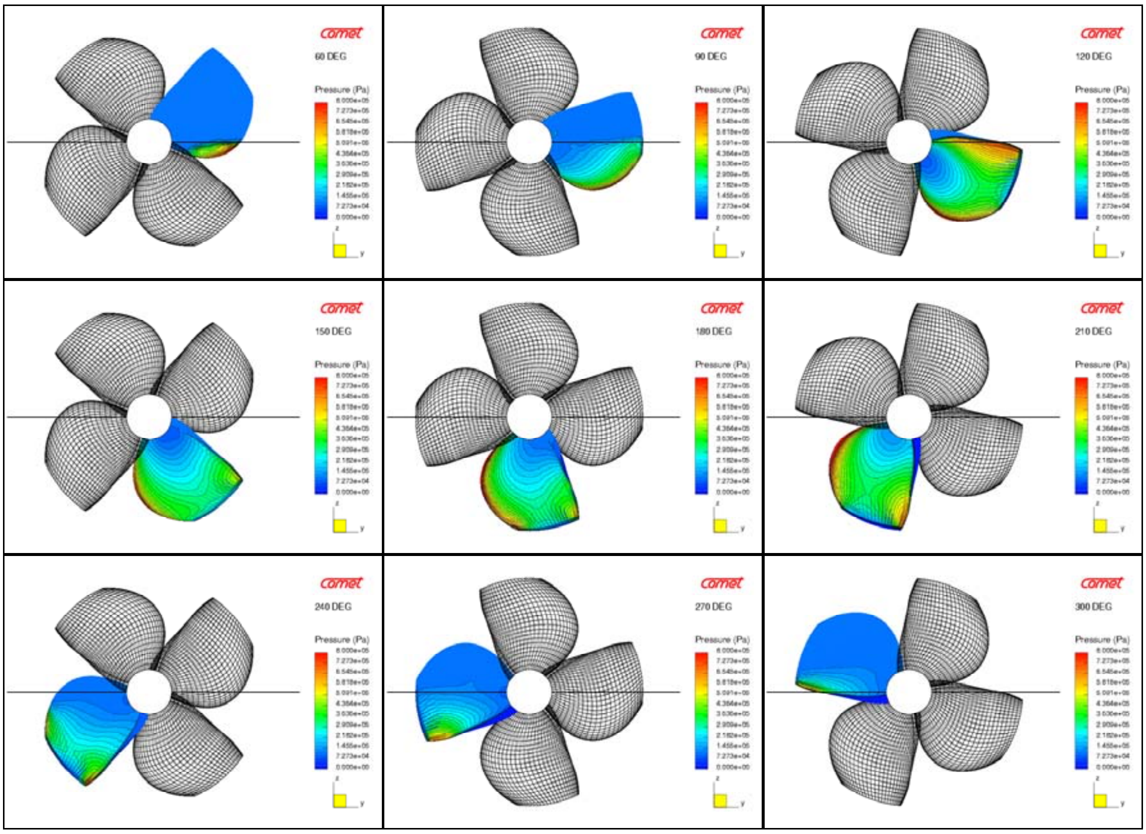
While in most of the cases performance calculations of submerged propellers can be carried on assuming steady conditions, the flow on an SPP is always unsteady and then computation must be solved marching in time, following the flow during some complete rotations of the screw.

In our approach the domain mesh is composed by two cylinders. A fixed external cylinder simulates the contours of the cavitation tunnel; on its lateral surface a slip condition is imposed, while on the fore and aft surfaces the inlet and outlet condition are imposed respectively. A smaller cylinder containing the propeller is set inside the external one. At each time step the internal cylinder is rotated of a small amount, and the propeller with it. Computational variables are interpolated at the sliding interface of the common surfaces of the internal and external cylinders. The axis of rotation of the internal cylinder can be oriented arbitrarily to represent the exact propeller shaft inclination.

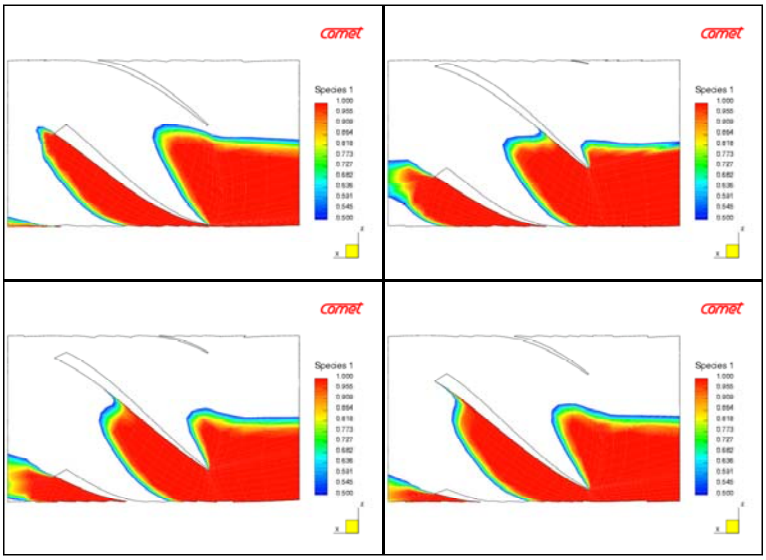
The Fortran code that built the mesh has been in house developed. The mesh inside each block is structured and uses hexahedral cells. Local refinements can be performed splitting a cell in a number of sub-cells. In the following examples the number of cells used varied from 200.000 to 500.000.

3. Flow visualization

Examples that could give more insight on the flow features of an SPP are presented in this paragraph. In the following series of figures the pressure distribution over the face of a blade is followed from the entry to the exit phases. It can be observed the pressure peak at the leading edge in the entry phase, but the maximum local pressure is found when the cup hits the water surface. In this moment also the thrust developed by the blade, and its mechanical stress, is maximal. When the blade is completely submerged the maximum pressure is along the leading edge (to promote ventilation) and along the trailing edge, especially close to the tip; in between the blade is relatively unloaded. When the blade is already above the undisturbed free surface level (last image), the cup is still loaded by the water that is pushed downstream but also upward; the large spray observed behind typical SPP is generated in this phase.

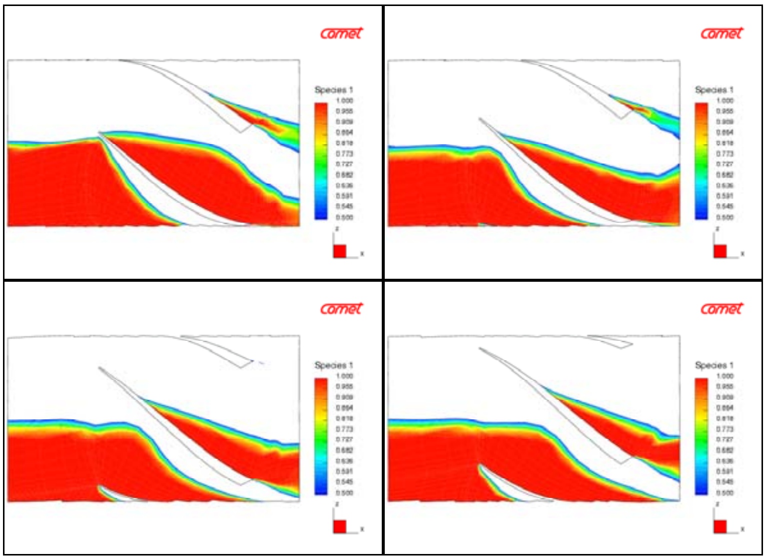


The next series of figures show a cylindrical section of the propeller at r/R=0.4. These images represent the cavity formation in the entry phase (flow from right to left). The cavity is unusually thick due the low advance ratio used for this calculation. It is very clear the influence between adjoining blades.



Entry phase

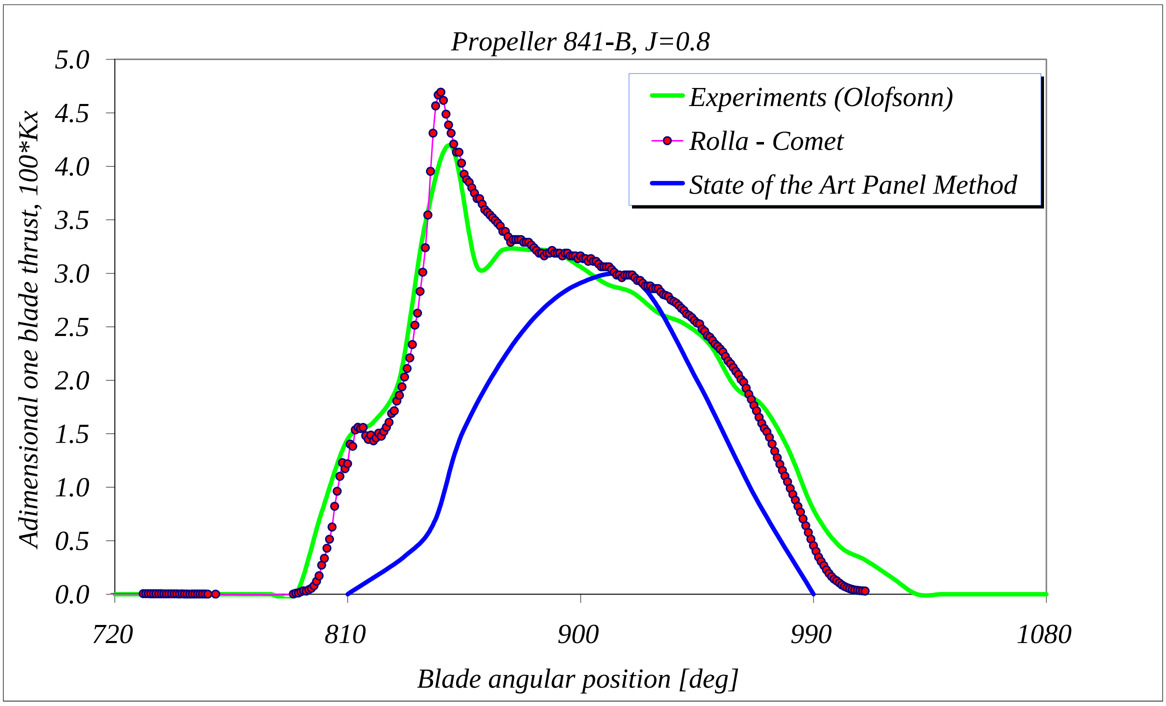
Similarly the last sequence of images shows the blades in the exit phase (flow from left to right). The profile approaches the free surface with the back side completely ventilated. The face carries with it some water even when it is well above the undisturbed free surface level.



Exit phase

4. Forces calculation

Despite flow visualization it is clear that our main goal is to be able to compute the correct forces and moments developed by the propeller. From our preliminary work the code seems to be able to supply the right answers even with a moderate numerical effort. The next figure shows a comparison between our computation and cavitation tunnel experiments. The x-axis represents the blade angular position while the y-axis is the adimensional value of the instantaneous axial force generated by one blade. The experimental data are from Olofsson. The agreement is very good. The main characteristics of the phenomenon are captured by the code, and the integral value of the thrust is computed with an error of few percents. Also shown is the result obtained by a panel method.



References

CAPONNETTO, M. (2000), *Numerical simulation of planing hulls*, 3rd Numerical Towing Tank Symp. (NuTTS), Tjärnö

CAPONNETTO, M. (2002), *Seakeeping of fast hard chine vessels using RANSE*, 5th Numerical Towing Tank Symp. (NuTTS), Pornichet

COMET, ICCM Institute of Computational Continuum Mechanics GmbH, Comet Version 2.004.A, Hamburg, now part of Computational Dynamics Ltd (CD Adapco Group)

DEMIRDZIC, I.; MUZAFERIJA, S.; PERIC, M. (1999), *Computation of turbulent flows in complex geometries, calculation of complex turbulent flows*, WIT Press

MUZAFERIJA, S.; PERIC, M. (1998*)*, *Computation of free surface flows using interface-tracking and interface-capturing methods*, Nonlinear Water Wave Interaction, Comp. Mech. Publ., Southampton, pp.59-100

OLOFSSON, N; Forces and Flow Characteristics of a Partially Submerged Propeller, PHD Thesis, Chalmers Tekniska Högskola, 1996.

ROSE, J.C.; KRUPPA, C.F.L (1991), *Methodical Series Model Test Results*, FAST'91, Vol. 2, Trondheim, Norway, June 1991.