Propelling marine applications

Mario Caponnetto, ROLLA Research, Switzerland

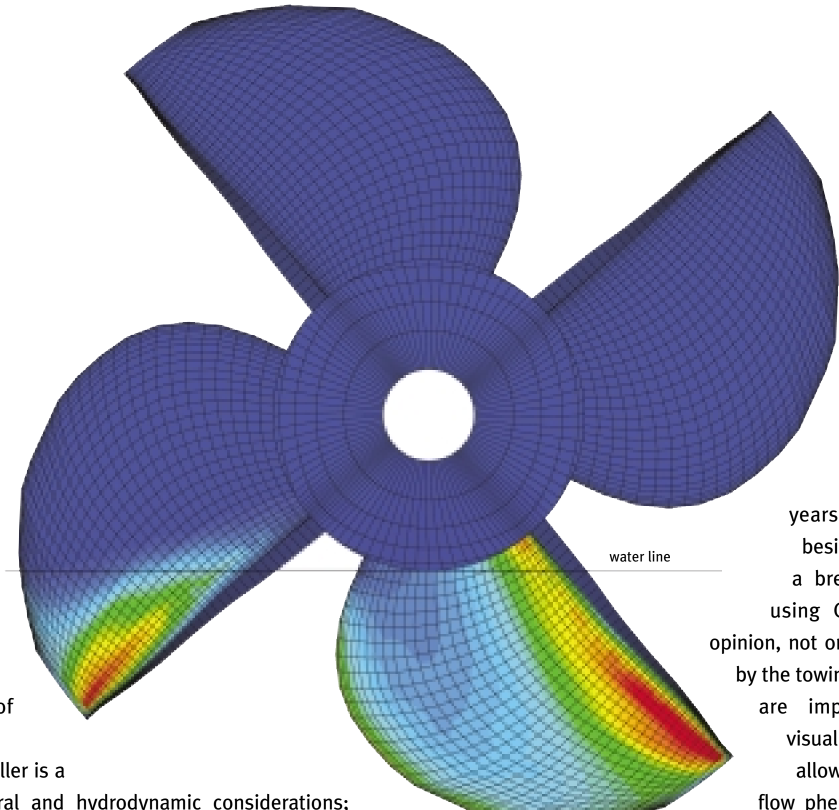


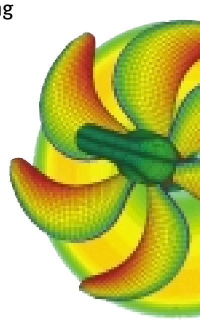
Fig.1: Calculated pressure distribution of a Suface Piercing Propeller

ROLLA SP Propellers SA was founded in 1980 by Philip Rolla in Switzerland. Since then the company has become the worldwide leader in the design and manufacture of high-speed propellers.

A properly designed propeller is a compromise between structural and hydrodynamic considerations; moreover it should match the boat and engine characteristics. An incorrect design may easily lead to performance and speed losses, as well as to cavitation, noise, vibrations and blade erosion. A clear understanding of propeller flow is therefore required. Firstly, it is necessary to know how the drag, trim and flow field around the boat change with speed; this should be done, in theory, with model testing in the so-called "towing tank". Second, once the propeller has been designed, it should be analyzed in model scale in a "cavitation tunnel". Both tests are time consuming and expensive (much more that the propeller itself in most cases) and for this reason they are seldom carried out for custom applications on small and medium size boats.

Research and innovation have always been the key factors of our success. In the past, a large effort has been undertaken by our company to test and optimize families of propellers in a cavitation tunnel. Some years ago it became clear that, besides experimental investigations, a breakthrough could be achieved using CFD. This is because, in our opinion, not only the raw numbers obtainable by the towing tank and the cavitation tunnel are important, but also the flow visualization offered by CFD programs allows better comprehension of the flow phenomena, thus speeding up the "learning curve" of the designer.

Our first objective was to use a tool capable of computing the hydrodynamic resistance of fast planing hulls, which represent the majority of our clients' applications. This is a field where state-of-the-art "Panel Methods" fail, mainly due to the complexity of the free surface generated (spray, breaking waves, complex flow at the transom stern). Among many commercial codes tested, we found that the CD adapco Group's alternative CFD solver Comet has best fulfilled our requirements. After a set up period, we obtained results of the same order of accuracy as a towing tank. This enabled us to understand the boundary conditions and the input data needed for propeller design. Moreover, we could look at the



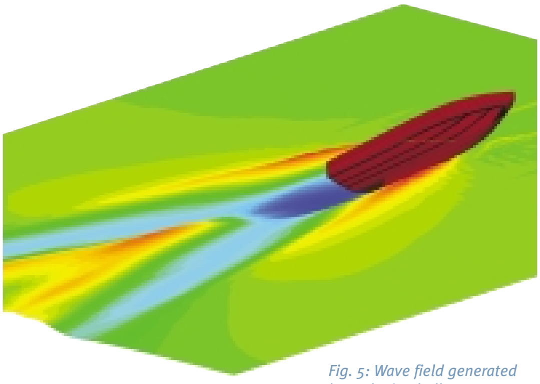
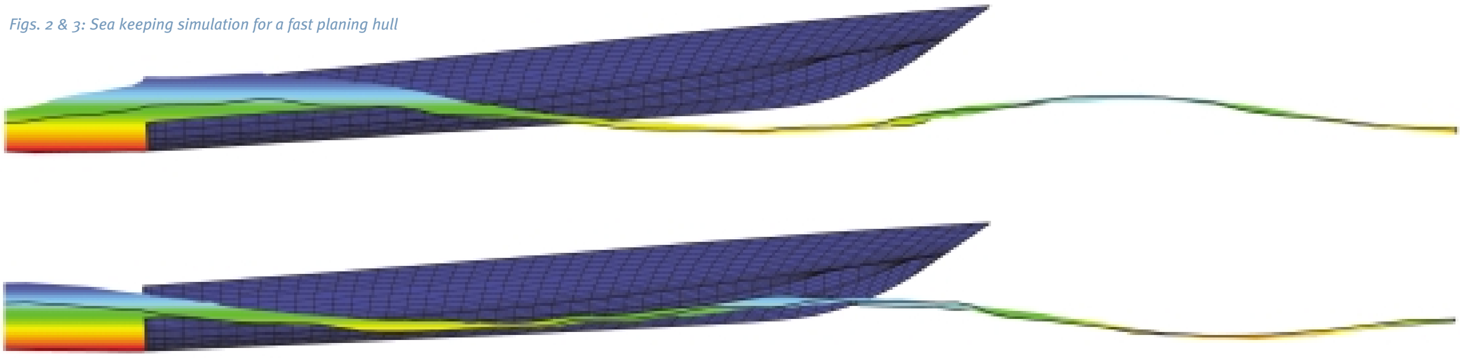


Fig. 5: Wave field generated by a planing hull

Figs. 2 & 3: Sea keeping simulation for a fast planing hull



pressure distribution and the flow field, detecting problems and suggesting possible improvements to designers of hull shapes.

Besides the computation of planing hulls in calm water, we found it useful to explore a similar field where, once again, no other tool was available for computation. Performances of planing boats are strongly affected by natural sea waves, and the best hull shape must be a trade-off between calm water and sea-keeping qualities. Sea-keeping simulations at any speed and wave characteristic, are now routinely performed using Comet. The most useful results are the level of acceleration onboard, the added resistance to waves and the instantaneous pressure distribution over the hull bottom, this last result being extremely useful for structure scantling.

Another field where the Comet solver proved to be a valuable tool was in the computation of Surface Piercing Propellers (SPP). At very high speed, when cavitation becomes unavoidable and strongly erosive, it is helpful to let the propeller work at the interface between the water and the atmosphere, allowing the water vapor cavities formed over the blades to be filled with air. In this case the free surface becomes extremely complex; a thin unsteady pocket of air is formed over the blade surface and a large spray is developed.

Using the VOF and "sliding surface" capabilities of the code, it has been possible to obtain results that match the data obtained in the cavitation tunnel within a few percent.

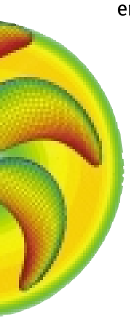


Fig. 4: CFD computation on a Submerged Propeller (velocity field)

Submerged propellers are designed using an "in house" developed Panel Method; this gives a reasonable approximation of the flow for conventional propellers working at the design point. Unfortunately, for off design conditions or heavily loaded propellers, unacceptably large errors may occur. For this reason every propeller designed is then checked in a wide range of conditions using the Comet solver and eventually corrected before manufacture. The large computational resources required for these CFD computations are achieved with an SGI Origin 2000 system, using 24 processors and 10 GB of RAM. An effort of flexibility has been essential to switch from the old established way of designing marine propellers to these new techniques; the ROLLA Research branch was created for this purpose. Its aim is to link engineers, production and customers, and provide them with the best tools for answering the many questions coming up in boat or ship design.

There is no doubt that these technologies have largely improved the quality of all our products, reducing the sources of error and uncertainty.

For further information, visit: www.rolla.com, www.rolla-propellors.ch