# Validation of Panel Methods for Propeller Flow Analysis

Mario Caponnetto, Rolla SP Propellers, Switzerland

Panel methods are considered a mature tool for propeller analysis and design. This is relatively true if comparing with other fields of marine application of panel methods, such as for predicting wave resistance or sea keeping. One physical reason is the relative smoothness of the flow that is achieved on marine propeller, compared with the complicated flow on the rest of the hull. From the numerical point of view, force computation using pressure integration is less sensitive of errors on the propeller than on the hull. But this accuracy is obtained mainly in open water conditions and around the design point. Results are in general less satisfactory when computing propellers at off design point, and even more unpredictable when including the effect of cavitation. In the present work a systematic comparison between open water measurements and computations is presented at first. The possibility to predict sheet cavitation is then discussed.

## *Open water computation*

The capabilities of numerical tools in predicting propeller performances are very attractive for propeller manufacturers when the kind of application doesn't allow the validation of the design with experimental methods, such as the towing tank or the cavitation tunnels. This is particularly true for small and medium size propellers, due to the obvious time and money constrains. Lifting line and lifting surface methods have been successfully used for a long time. Panel methods should represent an improvement with respect to lifting surface, due to the possibility to properly model the effect of blade thickness, as well as the presence of other thick bodies (hub, duct, and portion of the hull). While the computation of the global forces (and moments) should have an accuracy comparable with lifting surface, panel methods should allow a better determination of the pressure distribution, especially at the leading edge, and this is a valuable improvement for predicting cavitation inception. In its classical Morino formulation [1], panel methods are relatively easy to implement numerically, are fast and stable. The program developed in house at Rolla SP Propeller is a low order panel method, that uses a constant distribution of sources and doublets distributed over non-planar panels (PANAIR); a pressure Kutta condition is enforced at the geometrical trailing edge of the blade; the trailing wake is iteratively aligned to the local flow; viscous drag and viscous pitch reduction are implemented using empirical formula. The method has been extended from the original steady and non-cavitating original version to deal with the unsteady and cavitating cases. Moreover it is used as kernel of the design program currently used at Rolla SP Propellers for submerged propellers design [2].

It has been claimed that panel methods for propeller analysis are mature tools, namely can be applied for practical applications with accuracy comparable with experimental methods. It is true that, for a number of lucky coincidences, the accuracy for propeller analysis is much higher that for other marine applications, such has wave resistance and seakeeping computation. When facing the real life one have to be aware about this statement. A first attempt to validate systematically panel method for open water has been presented by the author in [2]. A total number of 36 B-series propellers, with different blade number (Z=4, 5, 6, 7), expanded area ratios (Ae/Ao=0.6, 0.8, 1.0) and pitch ratio (P/D=0.6, 1.0, 1.4) have been computed with the panel code in the whole range of advance ratios (J), for a total of 324 runs. The computed results have been compared with the experimental data and are presented in figures 1 and 2. While the scattering of the results is quite high, the most obvious trend is that the thrust is in general better predicted then the torque and particularly the torque is highly under-predicted at high loads (low J).

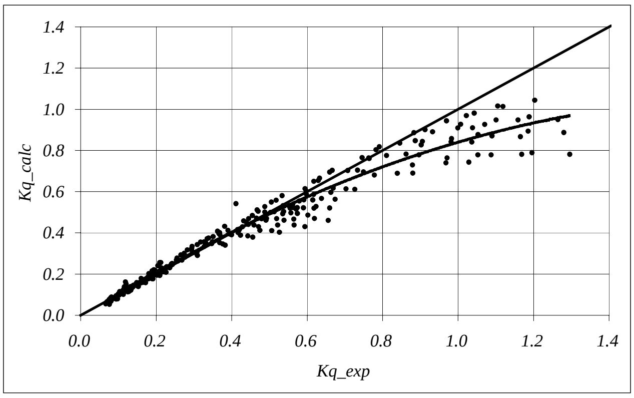


Figure 1

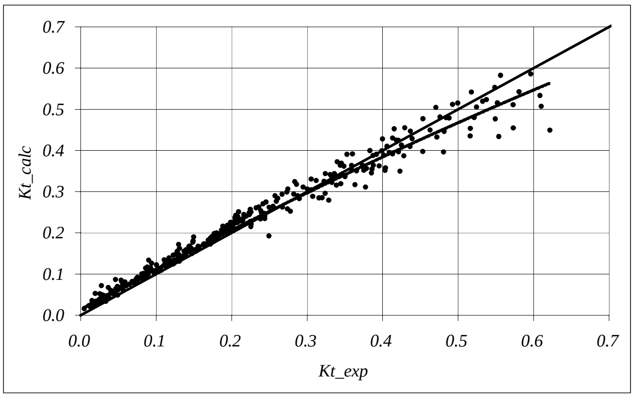


Figure 2

The B-series propellers have been used for this analysis since they represent the most complete and systematic series for propellers. As known, they have very simple blade geometry, with a constant pitch distribution, standard blade outline and skew, and "old-fashion" sections. In particular the profiles are quite different from those normally used nowadays to face cavitation problems, such as the NACA profiles, or even better those customised using for example the Eppler code [3]. It could be expected that for more complex geometries, typical of recent and sophisticated propellers, results should be worst then for the simple B-series; but may be that *simple* geometry doesn't mean necessarily *simple flow*. Indeed several propellers with more complex geometries have been analysed with the panel method, obtaining excellent comparisons with the experiments. As an example, this comparison is presented for propeller AO-177, designed for the US Navy. This 5 bladed propeller has 40 deg of skew, NACA profiles and a variable pitch and camber distribution designed to reduce the circulation near the tip and the hub. In figure 3 open water results for the Kt and Kq are compared with computations obtained with 3 different meshes.

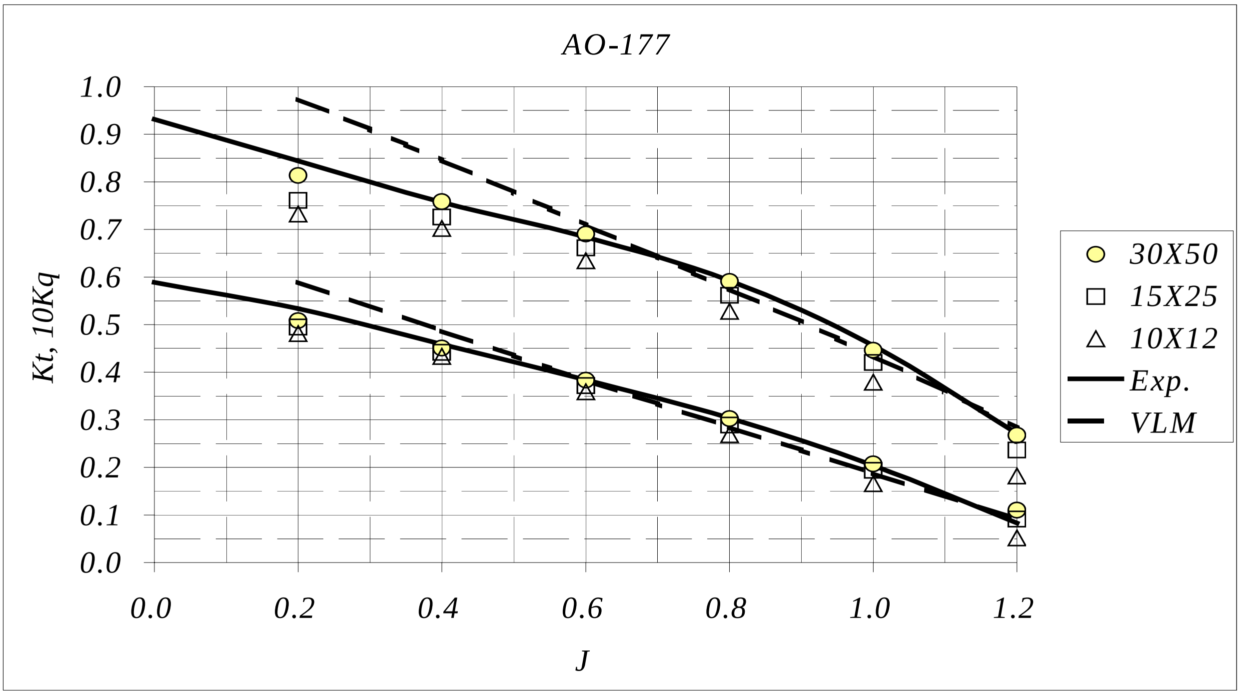


Figure 3

The finest mesh (30X50) consists of 30 spanwise equally spaced panels and 50 chordwise cosinusoidally spaced panels on both sides, for a total of 3000 panels per blade. The agreement with the experiments improves rapidly with the number of panels. The results obtained using a state of the art lifting surface program (VLM) is also shown in the figure. The results obtained with the panel method are extremely good in the range of advance ratios J=0.4/1.0; thrust and torque is slightly under-predicted only at J=0.2.

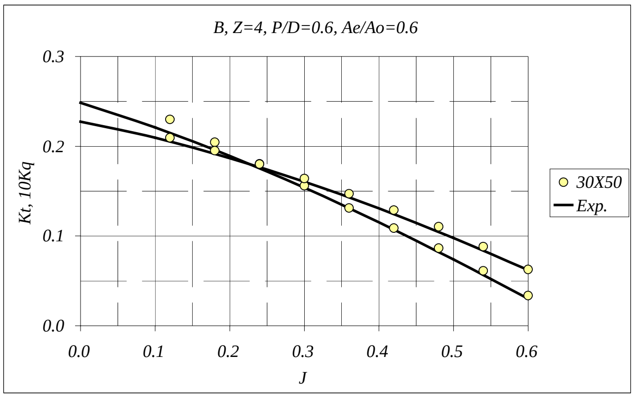


Figure 4

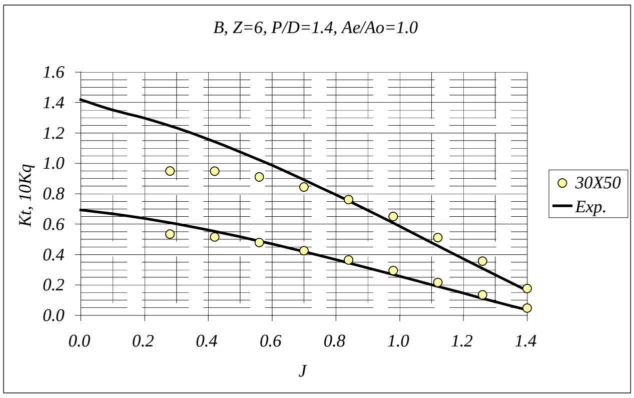


Figure 5

Another good result is presented in figure 4. The propeller is again a B-series, 4 bladed, with Ae/Ao=0.6 and P/D=0.6. At the opposite the propeller of the same family but 6 bladed, with Ae/Ao=1.0 and P/D=1.4 shows disappointing results for the torque below approximately J=0.7. In this case at J=0.28 the torque is under-predicted of about 24% and the thrust of 10%.

One of the possible explanations about the different accuracy of the panel method for different propellers could be explained with the effect of leading edge vortex separation. In the panel method the trailing vortex detachment is imposed at the geometrical cusped trailing edge of the blades. This hypothesis is reasonably satisfied as long as the angle of attack of the flow remains confined in a range of angle of attack above the ideal one. For swept leading edges, above a given angle of attack the flow may separate from the leading edge forming a free shear layer. This shear layer rolls up in a concentrated vortex (leading edge vortex) that flows over the suction side of the blade. The low pressure in the core of the vortex generates an extra lift that cause an increase of the propeller thrust and an even higher increase of torque. In principle, when operating at low J, a propeller with a reduced pitch at the tip, such as the AO-177 or the B-4-60 with P/D=0.6, should experience a lower angle of attack at the tip and consequently less leading edge separation with respect to the propeller with an high pitch (B-6-100 with P/D=1.4). In theory, knowing the position of leading edge separation, the effect of the leading edge vortex could be implemented in the frame of a potential flow method. A tentative approach has been proposed in [4] for a vortex lattice method. While this approach could be explored, it must be pointed out that, from a practical point of view, when the propeller is operating at J lower then the design point cavitation is quite always experienced, further complicating the local flow pattern.

## Cavitating propellers

Marine propellers in almost any kind of applications experiences cavitation. For large ship it is important to predict the cavitation pattern and volume variation, in order to predict noise and vibrations. For fast vessels it is mandatory to predict the variation of performances (thrust and torque), experienced by the propeller mainly due the inclination of the shaft. Panel methods can deal with stationary and non-stationary sheet cavitation; an outline of the method can be found in [5] and [6]. While the mathematical formulation of the problem is not particularly difficult, it has been pointed out that the computation of the cavitation pattern is very sensitive of the location of cavity detachment point, not easy to predict. In order to have an idea of the capability of the method for fast propellers, a comparison between computations and experiments performed in the cavitation tunnel is presented. In figure 6 the computed and observed cavity pattern is shown for the propeller operating at a fixed J and varying the cavitation number. The agreement is generally good. In figure

7 and 8 the computed and measured forces can be compared for two different J. The cavitation number of thrust breakdown is quite well predicted in both cases, but below this point a quite large discrepancy of the value of the forces can be observed.

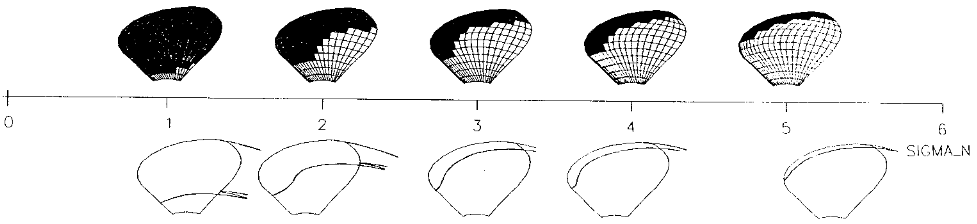
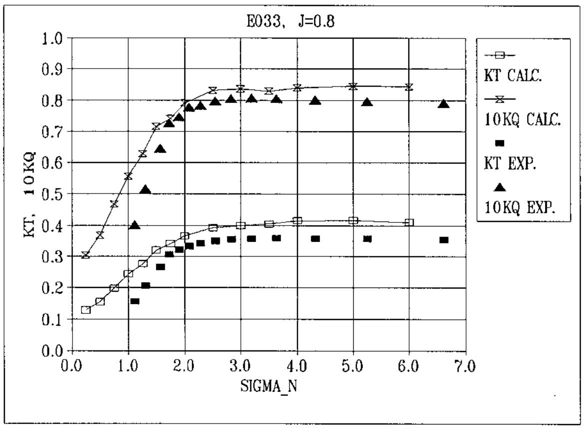


Figure 6



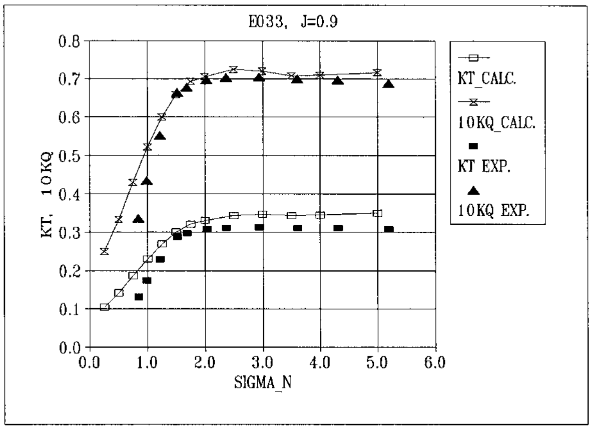


Figure 7

Figure 8

Another major problem experienced by fast propellers is cavitation and erosion at the junction between the blade and the hub when operating with a large inclination of the shaft. This phenomenon is difficult or impossible to model with a potential flow code, since the local flow is strongly influenced by the viscous flow coming from the brackets and the shaft. Moreover subtle changes in the geometry, such as leading edge or fillet radius, can have a large influence in this phenomenon. A few literature exists on this subject [7]; for this reason the author is trying to increase the knowledge using a Navier-Stokes solver and full-scale observation of the phenomenon.

## *Conclusion*

In this work a validation of panel codes for open water propellers has been attempted. A large number of propellers have been computed and compared with the available experimental data. It can be summarized that the results are in general satisfactory, for engineering purpose, for propellers working close to the design point, while noticeable discrepancies may exists at off design point. Most of this discrepancies could be probably ascribed to the difficulty of predict leading edge separation at large angles of attack. For what concerns cavitating propellers, the panel method used is generally capable to predict the condition of thrust breakdown, very useful for fast propeller design. The author suggests more research in the field of root cavitation for fast propellers.

## *References*

1. Morino, L & Kuo, C.C., “Subsonic Potential Aerodynamics for Complex Configurations: a General Theory”, AIAA Journal, vol.12 (no. 2), pp. 191-197, February 1974.

2. Caponnetto, M. & Rolla, P., “Theoretical and Practical Experience in Developing and Tuning Codes for the design and Analysis of Propellers”, CFD 99, 5-7 June 1999, Ulsteinvik, Norway.

3. Avellan, F. & Dupont, P., “Analysis of the Cavitation Behavior of Different Propeller Sections”, CTI N. 31016.1 Project, EPFL-IMHEF, Study done for Rolla SP Propellers SA.

4. Greeley, D.S. & Kerwin, J.E., “Numerical Methods for Propeller Design and Analysis in Steady Flow”, SNAME Transactions, Vol. 90, 1982, pp. 415-453.

5. Fine, N.E. & Kinnas, S., “The Nonlinear Prediction of Unsteady Sheet Cavitation for Propellers of Extreme Geometry”, VI Conference on Numerical Ship Hydrodynamics, Iowa City, 1993.

6. Caponnetto, M. & Brizzolara, S., “Theory and Experimental Validation of a Panel Method for the Analysis of Cavitating Propellers in Steady Flow”, Propcav 95, Newcastle upon Tyne, 1995.

7. Maioli, G. “Cavitazione delle Eliche nella Zona del Raccordo”, Annale N. 5, 1966, Ministero della Difesa, (in italian).