V International Conference on Computational Methods in Marine Engineering
MARINE 2013
B. Brinkmann and P. Wriggers (Eds)

COMPARISON OF DIFFERENT APPROACHES FOR THE DESIGN AND ANALYSIS OF DUCTED PROPELLERS

STEFANO GAGGERO[*], MICHELE VIVIANI[*], GIORGIO TANI[*], FRANCESCO CONTI[†], PAOLO BECCHI[‡] and FEDERICA VALDENAZZI[‡]

[*]Department of Naval Architecture, Electrical and Electronic Engineering
University of Genoa
Via Montallegro, 1, 16145 Genoa, Italy

[†] Fincantieri Naval Vessels Business Unit
Via Cipro, 11, 16129 Genoa, Italy

[‡] CETENA
Via Ippolito d'Aste, 5, 16129 Genoa, Italy

Key words: Decelerating Ducted propellers design, Panel Method, RANS,

Abstract. In the present paper, different approaches for the design and analysis of ducted propellers are presented and discussed, starting from the conventional lifting line / lifting surface approach and considering more complex (and computationally demanding) panel methods and RANS solvers. Attention is posed on the more challenging case of decelerating duct configuration, and a design case is presented for a thorough analysis of the various approaches. Two different propellers geometries have been defined, and the results of the experimental campaign at towing tank and cavitation tunnel carried out on them are shown, demonstrating the capabilities and limits of the adopted approaches. Finally, general guidelines for the design of this kind of propulsor are briefly outlined.

1 INTRODUCTION

Ship design requirements are always increasing in time, with new and challenging tasks for the designers, in order to be able to grant a higher quality and to face a very competitive shipbuilding market. Considering propeller design, the increasing requirements, especially for what regards the higher segment in the shipbuilding activity, focused on high added value ships, have led to the necessity of providing not only a propeller with high efficiency and avoidance of erosive cavitation, but with also good characteristics in terms of low induced vibrations and radiated noise. In the present paper, the activities carried out by Fincantieri, in cooperation with CETENA and the University of Genoa (DITEN), in the framework of the European Project BESST, are presented. In particular, attention has been focused on ducted propellers for medium/high speed applications and improved cavitation behaviour. As a consequence of this second requirement, decelerating duct configuration has been chosen, since they are thought to be an interesting alternative to free running propellers for ship propulsion, being potentially very performing in terms of reduced cavitation, vibration and noise. As it is well known, ducted propellers are currently mainly used in accelerating duct configuration for low speed applications where bollard pull is of great interest; in this case concepts and design methods related to these propulsors are well known since the early 70s [9], and many different works have been presented during years. On the contrary, the decelerating duct configuration has received lower attention, and design examples and/or experimental data are much more limited. This may be partly due to higher difficulties in the prediction of the propeller-duct interaction and, more in general, of the duct decelerating effect; as a consequence, in this case the propeller performance prediction is more challenging with the usual design and analysis tools. Recent developments in CFD provide new tools to support the design of ducted propellers and open new perspectives in exploiting their potential. In the activity carried out, aimed to the development of design guidelines for this kind of propulsor, different approaches for the design, optimisation and analysis of decelerating ducted propellers have been analysed. In particular, conventional lifting line/lifting surface approach [9], panel methods (coupled to genetic algorithms as optimizing tools [4, 1]) and RANS [2] calculations (in 2D and 3D) are considered. A brief description of methods considered is reported in section 2. In order to exploit the capabilities of these different approaches, a possible realistic design case has been considered, as described in section 3. At first, the design has been performed using the methodology usually adopted by Fincantieri for this kind of propellers [10]; this methodology, already applied with satisfactory outcomes, couples iteratively an in-house lifting line/lifting surface code (named "Elintub") and a commercial RANS solver (Ansys CFX). In order to further improve this procedure two more numerical methodologies have been investigated and described in this paper and concern the possibility of duct and blade geometry optimization. These different approaches rely on a quasi 2D actuator disk model for the duct design, analysis and optimization and a potential method coupled with a genetic algorithm for the blade geometry optimization. It is believed that these two approaches may be conveniently included in the design process, which of course includes, as represented in figure 1, towing tank and cavitation tunnel tests as final verification of mechanical characteristics and cavitating behaviour.

2 THEORETICAL BACKGROUND

2.1 Lifting Line / Lifting Surface Design Approach

The code, named "Elintub", used for the design activity described in this paper, is a lifting line/lifting surface based software, developed by Fincantieri and CETENA. As usual for a design tool, the definition of the propeller blade geometry is based upon the

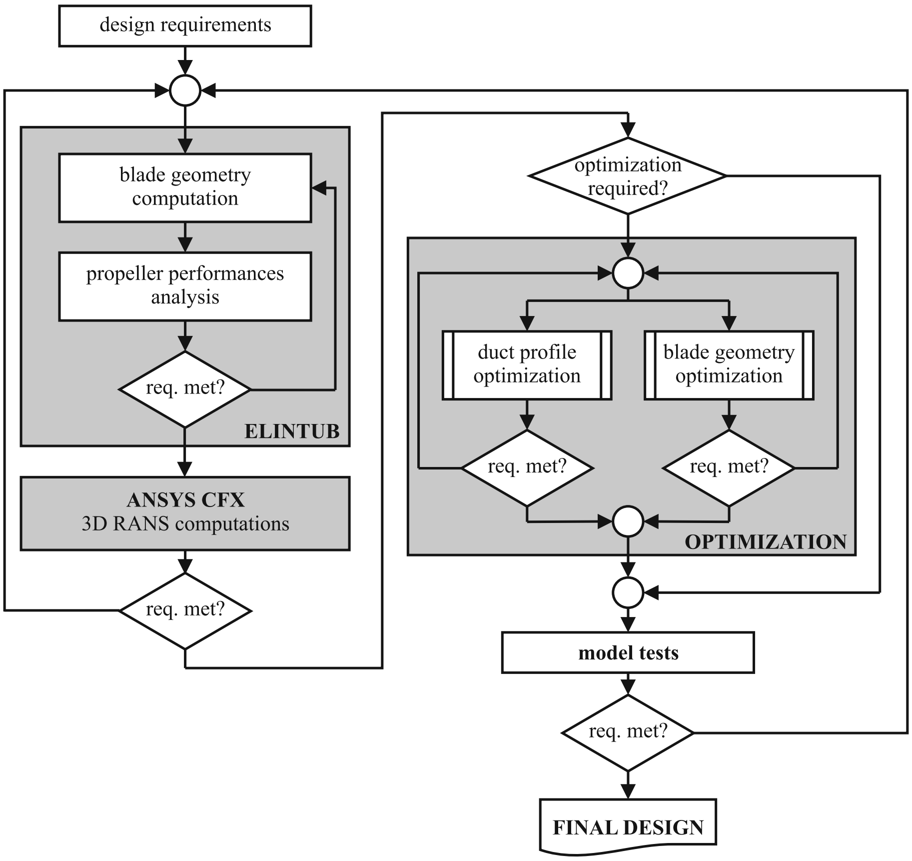


Figure 1: Ducted propeller design flow chart.

assumption of non-viscous, incompressible axi-symmetric and steady flow, with the blade represented by a lattice of vortexes, with unknown intensity and placed on an iteratively adjusted lifting surface. The influence of the duct is accounted through the linearized annular airfoil theory and is approximated by a distribution of ring vortices and sources. The design is carried out iteratively, until convergence is achieved in terms of the flow induced by the duct at the propeller and of the flow induced by the propeller at the duct.

2.2 Panel Method / Optimization by genetic algorithms

An alternative strategy to the classical lifting line/lifting surface approach is represented by optimization. The definition of a new geometry can be performed, in fact, testing thousands of different geometries, automatically generated by a parametric definition of the main geometrical characteristics of the propeller (as in [5, 4, 1]), and selecting only those able to improve performances (in terms of efficiency and cavity extension, for instance) together with the satisfaction of defined design constraints. The core of the design by optimization is represented by an accurate, reliable and fast flow solver and by a robust parametric representation of the propeller geometry. A potential panel method, such as the one developed at the University of Genova [6] complies the first requisite. The code has been specifically customized for the solution of cavitating ducted propellers with the inclusion of the tip gap flow correction as in [8]; a thorough description of the code may be found in [7]. With respect to lifting line/lifting surface approaches, a panel method allows to directly compute the influence of the hub and, especially, of the duct, both in terms of the additional load on the blade tip region and in terms of the velocity disturbance on the whole propeller, avoiding the simplified representation of the duct only by vortex rings and sources. The results accuracy versus computing time ratio is, moreover, extremely good (if compared with RANS solver) making this kind of solver suitable for the automatic analysis of thousands of geometries. The classical design table is the natural, robust, parametric description of the propeller geometry, that can be easily fitted by means of a set of B-Spline curves whose control points turn into the free variables of the optimization procedure. A genetic algorithm drives the optimization procedure: from an initial population (whose members are randomly created from the original geometry altering the values of the free parameters within prescribed ranges), successive generations are created via cross-over and mutation: the members of the new generations arise from the best geometries of the previous computations that satisfy all the imposed constraints (thrust identity, for instance) and grant better values for the selected objectives.

2.3 RANS 2D computations

The design of a ducted propeller requires the flow field around the blade tip area to be analyzed in detail, because viscous effects are known to have a primary role in what occurs in the gap between the blade and the duct, especially in terms of cavitation and then noise. As modelling a complete 3D RANS computations could be very time expensive in the design phase, both for meshing and computing needs, in the adopted design approach the complete 3D RANS computations are performed just in the advanced phase of the design procedure, in order to check an already well established blade and duct geometry and, if needed, give some indication to make the design closer to the requirements. However, in this way the whole flow field around the blade and the duct (and their interaction) can be known only at the end of the design procedure. For this reason, the feasibility of a simplified RANS approach has been studied and added to the traditional design procedure. The aim of this approach is the definition of a quick meshing and computing RANS simulation able to evaluate the flow field around the duct, its cavitation behaviour, its induced flow field on the propeller, in order to carry out a customized duct design/optimization. For this purpose, a quasi-2D actuator disk model has been developed. This kind of approach can lead to:

- very detailed mesh size around the duct, keeping the whole fluid domain size at least ten times smaller than a complete 3D computation,
- very fast and easy geometrical modification and re-meshing of the duct, making it possible to optimize the flow field around the duct by RANS simulation,

- estimate the induced flow field on the propeller and, in case, perform some modifications on the design input data in order to make the lifting line/surface evaluation closer to target needs.

The fluid domain is modelled considering a narrow slice of domain, with an angular extension of 2-3 degrees. The effect of the propeller is simulated inside the CFX solver by the actuator disk model, where the thrust and the torque distributions are provided by external sources (panel code or lifting line/surface code, for instance). This kind of approach can represent a very powerful optimization tool, especially in the case it is coupled with the panel method optimization tool, because both the blade and the duct geometry can be made closer to the design needs.

2.4 RANS 3D computations

The final step of the ducted propeller design is a fully 3D RANS analysis, able to accurately simulate the viscous phenomena that typically occur in the blade tip area. Despite the good results that can be obtained by the panel methods, the flow field between the blade tip and the duct is strongly characterized by viscosity and then both the velocity field and the cavitation behaviour can be properly evaluated only by viscous numerical computations. In the present work, the Ansys CFX code has been applied, considering two different meshes for evaluating the propeller mechanical characteristics and the cavitation behaviour.

3 DESIGN CASES

3.1 Ducted Propeller Design point

The propellers have been designed for the propulsion of a medium/high speed twin screw vessel at an advance ratio close to 1 and a cavitation index (σ_N) of about 1.5.

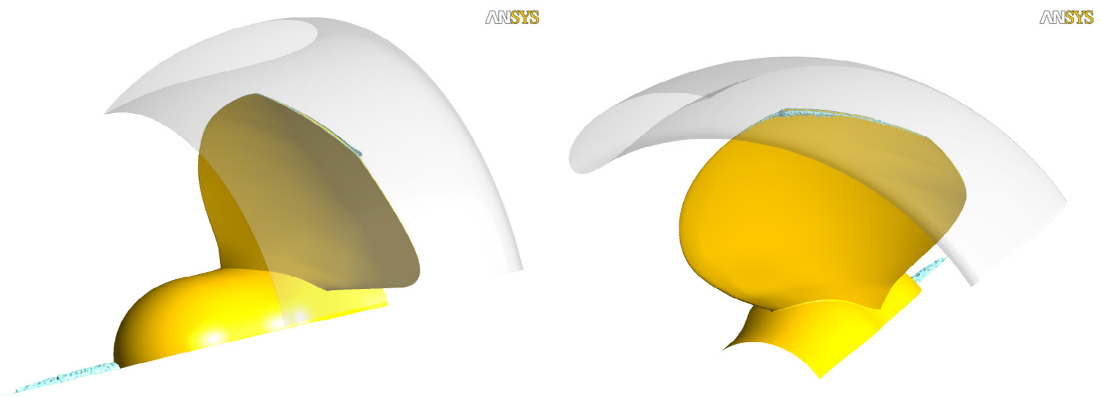
For confidentiality reasons the performance of the propeller will be provided in terms of thrust and torque coefficients (K_T and $10K_Q$) expressed as a percentage of the design coefficients.

3.2 Ducted Propeller Designed via Lifting Line/Lifting Surface and 3D RANS Computations - Propeller 1

As already remarked, the methodology usually adopted, with satisfactory outcomes, by Fincantieri, involves the coupling of the “Elintub” code, which evaluates the blade geometry meeting the design performance requirements, and CFX, which is able to investigate the propeller and the duct hydrodynamic behaviour. The analysis of the RANS results makes it possible to identify the modifications that have to be implemented in the design input data (load distribution, geometrical restriction, for instance) in order to optimize propeller performances. Usually a couple of iterations are necessary because the viscous phenomena, occurring especially at the blade tip and between blade and duct, strongly affect the propeller performances, especially in terms of cavitation inception and extension. During this work the RANS simulations used for the design have been set with a "coarse" unstructured mesh with 7.7 millions cells. Then, another "finer" mesh has been realized, using 15.6 millions cells. In particular, the flow domain geometry has been realized in order to focus the cells in the region close to the blade and the duct, consistently with the expected stream tube. The aim of this activity was the study of the effect of the mesh accuracy on the cavitation phenomena extension, on the pressure and forces results and the overall performances.

The two grids are adopted, respectively, to compute the overall propeller performance (K_T, K_Q, and η_o) and to check cavitation and appreciate the viscous phenomena in the gap between the blade tip and the duct. To this regard, it is necessary to remind that the computations have been carried out in "wetted" condition, i.e. without activating the cavitation (two phase flow) model; the cavitation pattern has been obtained at the design cavitation index by identifying the blade/hub areas where the pressure is lower than the vapour pressure. For what regards mechanical characteristics, the predicted propeller thrust in the final iteration is 99% of the design value. As it will be seen, this prediction has been verified by experimental results, thus confirming the validity of the adopted approach.

Figures 2(a) and 2(b) show the cavitation pattern from the computations, highlighting the presence of tip leakage vortex and hub vortex; also in this case, experimental results confirmed numerical predictions.



(a) Propeller 1 - Pressure side (b) Propeller 1 - Suction side

Figure 2: Cavitation on Propeller 1 by isosurfaces at $C_p = -\sigma$.

3.3 Ducted Propeller optimized by genetic algorithm - Propeller 2

Propeller optimization has been carried out in order to obtain a new geometry (Propeller 2), able to maximize efficiency and to reduce the back cavitation at the design cavitation index delivering the same numerical thrust of the initial propeller (the Propeller 1): in order to speed up the convergence, a thrust variation of $\pm 2\%$ is admitted. Propeller 1 design table has been used to define the range of the free parameters. The numerical predictions of thrust and torque obtained with the panel method for the Propeller 1 showed some differences with respect to the available experimental measures carried at towing tank (and with respect to the viscous computations previously presented); namely, total propulsive thrust was overestimated by about 7% and torque by about 12%. For the optimization it has been assumed that these differences, ascribed to the numerical approach, remain the same also for the newly designed propellers and the numerical predictions for the Propeller 1 have been taken as the reference point of the optimization procedure. Also in terms of cavity extension some limitations of the panel approach have to be be highlighted. The previous experience with Propeller 1 at the cavitation tunnel showed, at the design point, only a cavitating tip leakage vortex (plus less significant hub vortex) whose prediction is beyond the capabilities of the cavitating panel method. The sheet cavitation, that has been numerically evidenced at the blade tip of Propeller 1 (figure 7) can be, however, correlated with the occurrence of the tip cavitating vortex and its extension (to be, as a consequence, minimized) can be considered a measure of the risk of cavitating tip leakage vortex. In order to numerically amplify the sheet cavity bubble at the blade tip, to include a certain margin for the occurrence of bubble cavitation and let the optimization work at a more convenient point (for which cavity extension is not constrained by the dimension of few panels at the blade leading edge), the design of the new propeller via optimization has been carried at a slightly lower cavitation index with respect to the design point.

The optimization activity for the design of the new geometry has been carried out investigating only global parameters, i.e. maintaining the blade and duct profiles shape adopted for Propeller 1. In particular (control points of) chord, maximum camber and pitch distributions along the radius have been taken as free variables. Structural considerations have been limited, in present activity, to constraining the maximum thickness to the chord distribution in order to maintain the same blade strength of Propeller 1. About 10 thousands different geometries have been generated and analysed by the panel method, and the results of the optimization are reported in the Pareto diagram of figure 3. The optimal selected propeller, as highlighted in the diagram, is a compromise between reduction of computed cavity area and increased efficiency. As shown in figure 3, with respect to the computed values of efficiency and cavity area of Propeller 1, the new geometry, at the same working point, presents, numerically, a reduction of the cavity extension of about 30% and an increase in efficiency of 2%. Delivered total thrust of the new Propeller 2 is 1% lower than that computed for the Propeller 1 but, as expected, within the prescribed numerical tolerance of $\pm$ 2%. Reductions in cavity extensions only affect the cavitating strip of panels at the blade tip, whose significance (and correlation) in terms of cavitating tip leakage vortex would merit a deep investigation. The chosen propeller represents an optimal choice for the verification purpose of the reliability of the

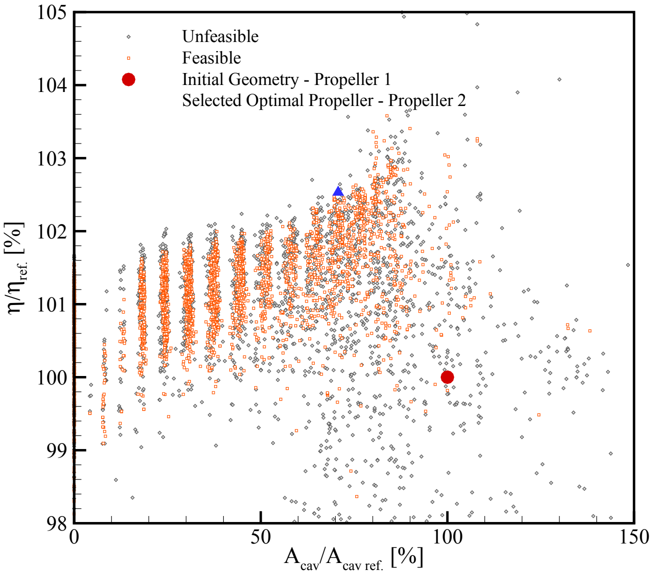


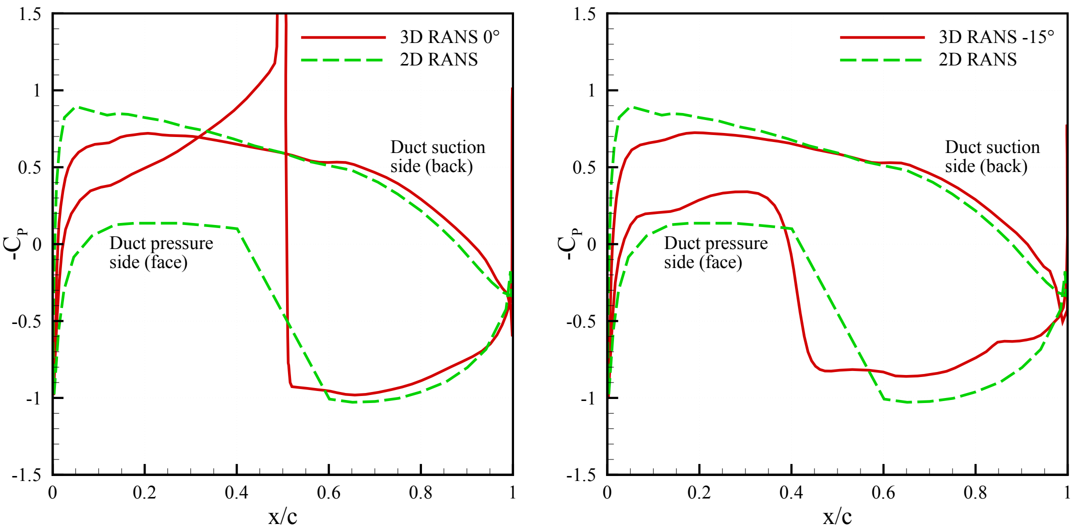
Figure 3: Pareto distributions of optimal geometries.

optimization approach, since efficiency can be better estimated and compared than the cavitating tip vortex strength; moreover, the design assumption about the risk of bubble cavitation may be also verified.

3.4 Application of 2D RANS computations in the optimization loop

In order to check the applicability of the quasi-2D approach described above, a test has been made considering Propeller 1. In particular, an actuator disk has been built based on the results from the 3D RANS computations of the propeller itself. Specifically, the actuator disk has been loaded with the thrust and the torque distributions computed for the propeller inside the duct, in a way to obtain the same total thrust and torque of the 3D computations. The computation results have been compared with those of the full 3D RANS model; this has provided the means to assess the reliability of using a quasi-2D approach instead of a full 3D approach.

Figure 4 shows the comparison of the pressure coefficients ($-C_P$) on the pressure side (face) and on the suction side (back) of the duct. Because the 2D simulation is characterized by axial symmetry, the pressure coefficient distribution over the duct profile is compared with what obtained by the 3D analysis, considering two different positions, the first one in correspondence to the blade passage, the second one in correspondence to 15 degrees toward the propeller leading edge. As it can be seen, a fairly good agreement is found in the second case (figure 4(b)), both for the back and the face of the duct. Otherwise, when the blade passage position is considered, it may be observed that (C_P) curves have a good agreement on the back of the duct while on the face, the pressure distribution from the 3D calculations features a sharp peak, which is associated to the flow field passing through the blade-duct clearance and then to the tip leakage vortex detected by the complete 3D computations. In the same longitudinal positions of the peak, the pressure distribution from 2D calculations features a linear pressure variation due to the presence of the actuator disk; the peak is absent, as the actuator disk cannot take such effect into account and also because the body forces characterizing the actuator disk are distributed over the whole disk volume. Since the total thrust (and torque) applied by the actuator disk on the flow is the sum of all the forces acting on all the cells of the disk volume, the effect of the propeller increases linearly moving along the shaft. In any case downstream of the propeller the pressure distributions are satisfactory close each other.



(a) $\theta = 0^o$ (b) $\theta = -15^o$

Figure 4: Comparison between 2D and 3D computed duct pressure distributions.

As a result of this analysis, it can be said that, as far as the global duct behaviour is concerned, the comparison between 3D and quasi-2D is good and makes quasi-2D computations a feasible alternative to 3D ones, especially in the propeller and duct design phase, allowing to capture the global functioning characteristics of the duct. Of course, effects such as tip leakage vortex and the local accelerating effect of the blade, are not captured by quasi-2D computations, as visible, and fully 3D computations need to be carried out at some point in the design process to account for these effects. As a conclusion, it can be said that quasi-2D RANS computations can be used to investigate the duct behaviour, provided that data are available to set-up the actuator disk. Considering both the meshing and computing time needs, the 2D model looks very attractive as a design tool (coupled with lifting line/lifting surface computations) because it makes it possible to investigate in detail the hydrodynamic behaviour of the duct profile, consistently with the load distribution characterizing the propeller blade. Furthermore, this kind of approach can increase in efficiency in the case of it is coupled with a propeller analysis and optimization tool, such as that described in the previous section. However, full 3D computations need to be carried out if information on cavitation and local effects in the blade tip area are to be predicted with care.

4 EXPERIMENTAL CAMPAIGN

4.1 Experimental Setup

Model tests (open water tests and cavitation tunnel tests) have been performed in order to validate the numerical results and the adopted design procedures.

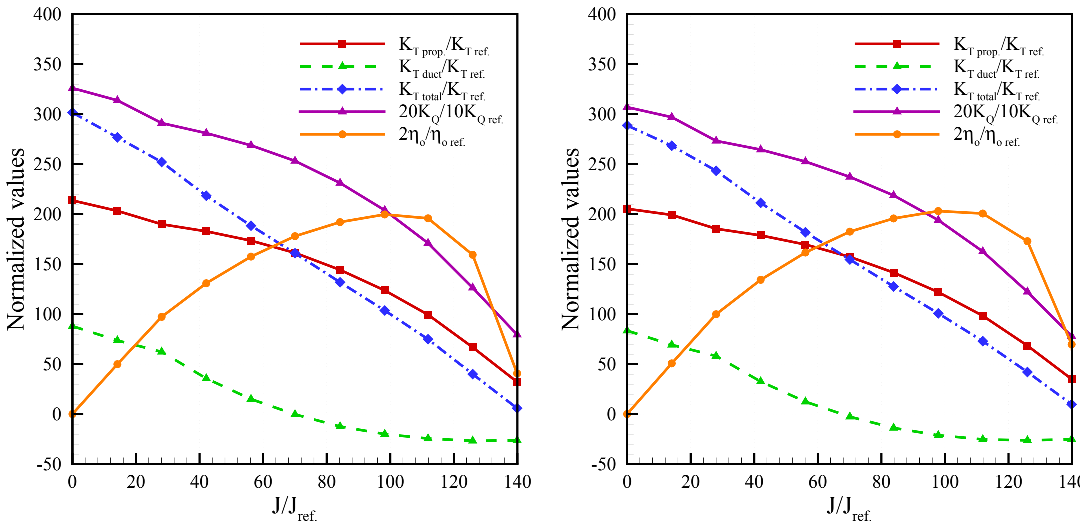
In particular, open water tests have been carried out at SVA towing tank, using a Kempf & Remmers propeller dynamometer H39 and a R35X balance for the measurement of duct thrust. A constant propeller rate of revolution (15 Hz) was adopted during tests. Cavitation tunnel tests have been carried out, instead, at the University of Genoa cavitation tunnel. The tunnel is equipped with a Kempf & Remmers H39 dynamometer, which measures the propeller thrust, the torque, and the rate of revolution. As regards the duct forces, an in-house developed measuring device has been adopted, allowing to perform not only usual cavitation observations, but also direct measurements of different forces components (and thrust breakdown, if present) in cavitating conditions. A detailed description of the measuring device may be found in [3]. All tests were carried out without propeller shaft inclination and in an uniform wake, consistently with the design assumptions previously described. A constant propeller rate of revolution (25 Hz) was adopted.

4.2 Open Water tests

Results from model scale open water tests carried out at SVA towing tank are shown in figure 5. The reported values are normalized with respect to the design point. Open water tests substantially confirm the reliability of the two different adopted design procedures, showing for both the propellers a good agreement with the required design performances; Propeller 1 has a thrust coefficient almost equal to the required one, confirming the numerical results presented in section 3.2 while Propeller 2 delivers, in line with the numerical results of the optimization, a little lower thrust (about 3%). On the other hand, as expected, Propeller 2 presents a slightly increased efficiency (about 2%), confirming from this point of view the reliability of the optimisation procedure.

4.3 Cavitation observations

The aim of the cavitation tunnel tests has been the determination of the cavitation bucket for the two propellers, as well as the collection of a set of cavitation observations at a number of functioning points in order to validate the numerical predictions. The

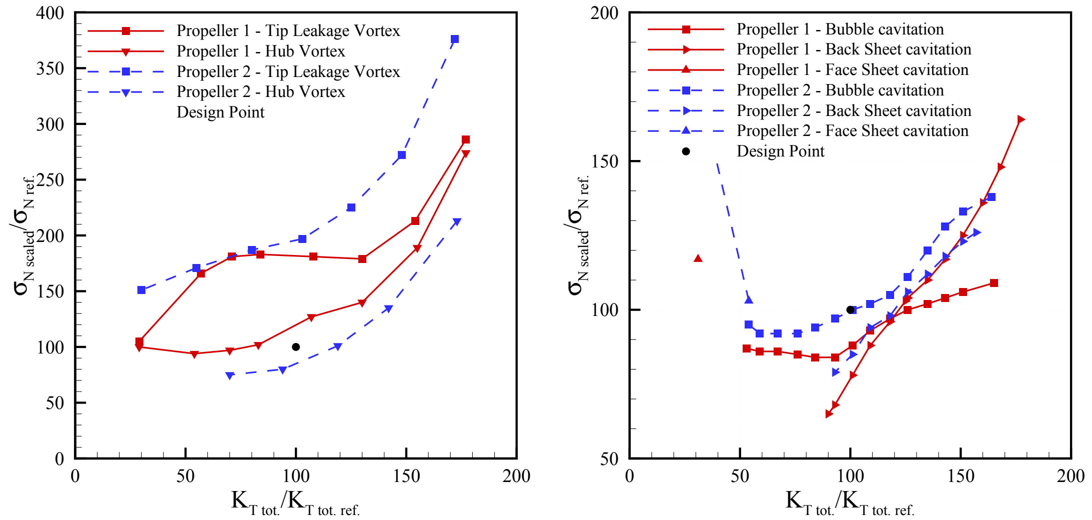


(a) Propeller 1

(b) Propeller 2

Figure 5: Normalized Open Water Performances.

inception points of the various cavitating phenomena are reported in figure 6(a) for what regards the vortex type occurrences (tip leakage and hub vortex), while in figure 6(b) Propeller 1 and Propeller 2 are compared in terms of the inception of the bubble and sheet cavitation related phenomena. The two propellers showed a satisfactory cavitating behaviour. At design point (figure 7(a) and 7(b)) tip leakage vortex and hub vortex are present for the Propeller 1 while only tip leakage affects the hydrodynamic behaviour of Propeller 2. However, this phenomenon is slightly anticipated, contrarily to what expected from the optimisation process, with respect to Propeller 1. This unwanted behaviour is probably due to the intrinsic limitation of the code, which does not allow to correctly rank the two propellers in terms of tip vortex inception merely by the predicted sheet cavity bubble at the last strip of panels at tip. From this point of view, it is believed that a final verification by means of RANS codes (including cavitation prediction) of a series of selected designs may overcome this problem, allowing to accurately characterize the propellers in terms of cavitating tip vortex inception and strength. The tip leakage vortex is present at any loading condition and its inception index seems less influenced by the propeller load if compared to the inception of the conventional propellers tip vortexes: in particular for Propeller 1 this phenomenon does not depend on the thrust coefficient for a wide range of values around the design point. This quite different behaviour of the tip leakage vortex inception (together with a different margin for what regard bubble cavitation at tip trailing edge) is probably the most clear difference between the propellers in cavitating conditions. For both the propellers, however, the buckets are quite wide, therefore confirming the capability of the decelerating ducts to postpone cavity inception also in off-design conditions and the general capability of both the design strategies to provide satisfactory geometries.



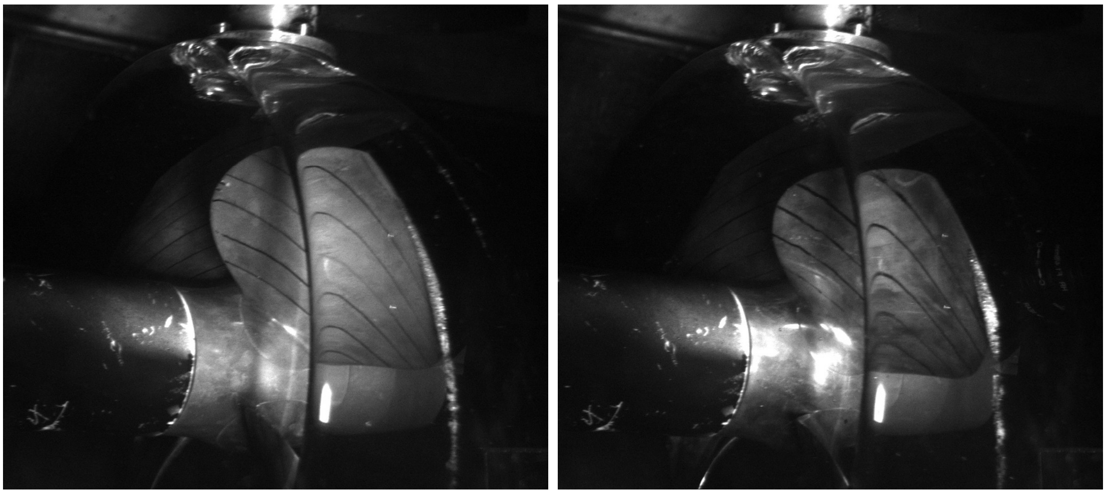
(a) Vortex phenomena (b) Bubble phenomena

Figure 6: Normalized Cavitation Buckets.

The propellers look also free from bubble cavitation, which appears only at cavitating indexes lower than the design point. Margins, however, are very different. Propeller 1 is quite safe with respect to this phenomenon, while Propeller 2 experiences tip back bubble cavitation at the blade trailing edge just below the design point, as clearly visible from figure 6(b). These differences partially confirm the numerical calculations. Even if, as already mentioned, during the optimization activity a certain margin has been adopted (carrying out the new design at a cavitation index 10% lower than the design point), the analysis of the pressure distribution for a radial section near to the tip shows a pressure distribution at midchord closer, for Propeller 2, to the cavitation limit, confirming that the optimized propeller is more inclined to be subjected to bubble cavitation with respect to Propeller 1. Reasons for the discrepancy between predicted and observed bubble cavitation inception point have to be further investigated in future studies.

5 CONCLUSIONS

The present work shows the design of a decelerating ducted propeller, performed by a lifting surface code, checked and tuned by RANS calculations and optimized by a potential method coupled with a genetic algorithm. While the main design activity represents the traditional shipyard design methodology, that is the use of a own lifting surface design code coupled with some RANS simulations, the propeller geometry optimization was aimed to check the possibility of improve the design and make it closer to design requirement. Then, this study have led to two propeller geometries, the first obtained by the



(a) Propeller 1 (b) Propeller 2

Figure 7: Observed Cavitation for Propeller 1 and 2 at design point.

traditional shipyard design tool the second by the numerical optimization. The reliability of numerical codes adopted has been validated by means of the experimental campaign at towing tank and cavitation tunnel. Furthermore, the hydrodynamic behaviour of the duct has been studied by a quasi-2D RANS method, based on the actuator disk model. Considering both the meshing and computing time needs, this method looks very attractive because it leads to accurate information keeping the time restricted and consistent to the design time requirements. As expected, the application of the shipyard traditional design procedure has led to a good agreement between numerical prediction and experimental data; however the analysis performed by a panel method coupled with a optimization algorithms has shown to be very powerful and effective, making it possible to modify the propeller geometry in accordance with design restrictions and target. Nevertheless, panel methods presented an intrinsic limitation in completely correctly capturing phenomena which are characterized by a predominant viscous nature; in particular, despite cavitating tip leakage vortex occurrence is somehow predicted by the presence of cavitating panels at tip, the code is not capable of ranking different propellers correctly. From this point of view, therefore, it is believed that RANS calculations including cavitation may provide a better insight in the phenomenon, overcoming the problem and allowing to choose among a set of possible optimal designs the best solution including also design characteristics which may not be completely captured with less accurate approaches.

6 ACKNOWLEDGEMENTS

The research leading to these results has received funding from the European Community's Seventh Framework Programme (FP7/2007-2013) under grant agreement n 233980.

REFERENCES

[1] Becchi, P., Grasso, A., Bruzzone, D. and Valdenazzi, F. An integrated CFD-based tool for propeller blade design and evaluation. *International Conference on marine Research and Transportation, ICMRT 2005, Ischia, Italy* (2005).

[2] Becchi, P. and Pittaluga, C. Comparison between RANS Calculations and Panel Method Results for the Hydrodynamic Analysis of Marine Propellers. *MARINE CFD 2005 Conference, Southampton, England* (2005).

[3] Bertetta, D., Bertoglio, C., Conti, F., Rizzo, C.M. and Viviani M. Cavitation Tunnel Tests on Ducted Propellers. *17th International Conference on Ships and Shipping Research, Naples, Italy* (2012).

[4] Bertetta, D., Brizzolara, S., Gaggero, S., Savio, L. and Viviani M. CPP propeller cavitation and noise optimization at different pitches with panel code and validation by cavitation tunnel measurements. *Ocean Engineering* (2012). **53**:177–195.

[5] Gaggero, S. and Brizzolara, S. Parametric CFD Optimization of fast marine propellers. *10th International Conference on Fast Sea Transportation, Athens, Greece* (2009).

[6] Gaggero, S. and Brizzolara, S. A panel method for trans-cavitating marine propellers. *7th International Symposium on Cavitation, Ann Arbor, Michigan* (2009).

[7] Gaggero, S., Brizzolara, S. and Villa, D. Design and Analysis of Conventional and Ducted Propellers: a Numerical Approach. *17th International Conference on Ships and Shipping Research, Naples, Italy* (2012).

[8] Hughes, M.J. Analysis of multi-component ducted propellers in unsteady flow. *Ph.D. Thesis, Massachussets Institute of Technology* (1993).

[9] Oosterveld, M.W.C. Wake adapted ducted propellers. *MARIN Pubblication 345* (1970).

[10] Valdenazzi, F., Sebastiani, L. and Becchi, P. CFD based design oriented tools at CETENA. *International Conference on Computational Methods in Marine Engineering, Oslo, Norway* (2005).