From Single to Multistage Marine Propulsor: A Fully Numerical Design Approach

Stefano Gaggetto¹, Davide Grassi², Stefano Brizzolara¹

¹Marine CFD Group, Department of Naval Architecture, Marine and Electrical Engineering, University of Genova, Italy.
²ZF Marine Group, Arco di Trento, Italy.

ABSTRACT

The limitations of classical theories for the design of contra-rotating propellers suggest the necessity to develop fully numerical procedures to address designs with different front/rear rotational speed and/or number of blades, as well as unbalanced distributions of thrust between the propellers and the presence of hub and ducts. The present paper compares the design of a couple of contra-rotating propellers carried out by a classical Morgan-Lerbs approach and a fully numerical procedure based on a variational approach (Coney 1989) for the bounded circulation. Results in terms of circulation distribution, self-induced velocities and blade geometry are presented. Three different analysis tools, a lifting surface, a panel method and a RANS solver are finally applied to analyze the hydrodynamic performance of the generated designs in order to validate the design approach. It will be shown that the propellers designed with the two proposed approaches satisfy the design requirements as confirmed by all the three analysis methods. Moreover, it will be stated that the balanced load solution is the more efficient.

Keywords

1 INTRODUCTION

Contra-rotating propellers have been widely used for decades in outboard propulsion units for fast-planing craft or in podded drives for fast ships or mega-yachts due to its advantage in terms of overall propulsive efficiency, mechanical balance and shallow draft (an important issue, in particular for pleasure boats). By splitting the thrust and torque between the fore and aft propeller, the expanded area ratio and the diameter can be reduced, keeping the same cavitation margin when compared to a classical single propeller solution. However, the potential application of contra-rotating propellers is really wide, also being suitable for large displacement ship in conventional shaft line arrangement. The present paper deals with comparison between two different design approaches and three analysis tools; a set of contra-rotating propellers for podded drives has been designed through a classical lifting-line theory and a fully numerical lifting-line approach; and then the resulting geometries have been analyzed via a lifting-surface code, a panel code and commercial RANS tool in order to compare performances in terms of thrust/torque curves and induced velocities. The classical lifting-line approach is based upon the Morgan-Lerbs theory for calculating optimum blade circulation with some additional features like the numerical treatment of the induced velocities, the possibility to unload the optimum blade circulation at tip and hub radial positions and a fully automated blade geometry optimization for cavitation and strength assessment. While the foregoing method assumes a thrust/torque balance between the front and aft propeller and does not take into account for any hub effects, the fully numerical design approach solve a Lagrange multiplier minimization problem allowing for different load distribution and modelling the presence of the hub.

For the analysis, a lifting surface (Grassi & Brizzolara 2009) and a panel (Gaggetto & Brizzolara 2009) code, both developed for single propellers, have been modified in order to account for multi-stage propulsor, following an iterative approach and solving each propeller as a single one in a wake-adapted condition where the inflow velocity is calculated by solving the other propeller. Moreover, in order to solve both the propellers together, a RANS commercial solver has been used following a quasi-steady and a fully unsteady approach. The former RANS solution approach allows for a fast averaged solution of the contra-rotating propeller set, each operating in a time constant inflow wake. In this sense, the quasi-steady approach is similar to the iterative approach based on the mean inflow, adopted for the analysis of the contra-rotating propellers by the potential flow based methods. On the other hand, the RANS fully
unsteady solution allows to compute the unsteady effects related to the time varying relative positions between front and rear propellers and to have a better insight into the hydrodynamic phenomena that characterize contra-rotating propellers.

2 DESIGN PROCEDURE

As mentioned before, the present work uses two different approaches for determining the optimum blade circulation, whereas a common method (see Chapter 2.3) is adopted for generating the blade shape.

2.1 Revised Morgan-Lerbs Approach

The first design approach (Brizzolara et al. 2008) is based upon a revisited Morgan-Lerbs theory (Morgan et al. 1960). The design theory describing contra-rotating propeller is more complicated with respect to that for the single case, due to the interference between the propellers itself; for this reason, some simplifying assumptions are needed in order to come to a partially analytical solution of the mathematical problem. The velocity diagram for fore and aft propeller is depicted in Figure 1.

The key point lays in the definition of self-induced and interference velocities. Under some assumption regarding the induced velocities that comes from the momentum theory, Morgan derives the following relationship between self-induced and interference velocities:

\[
\begin{align*}
(u_{ai})_1 &= (u_{ai})_2 f_a [1 - (g_a)_2] \\
(u_{ti})_1 &= 0 \\
(u_{ai})_2 &= (u_{ai})_1 f_a [1 - (g_a)_1] \\
(u_{ti})_2 &= 2(u_{ai})_1 f_t [1 - (g_t)_1]
\end{align*}
\]

Where subscripts 1 means at forward propeller, 2 means at rear propeller, a stands for axial, t for tangential component, s for self-induced and i for interference induced components of velocity.

The foregoing procedure for unloading the blade circulation will also be applied to the fully numerical approach described later. Once the circulation for the Equivalent Propeller is derived, the next design stage is to calculate the hydrodynamic pitch for the actual propeller by introducing some longitudinal interference factors, \(g_a\) and \(g_t\) (being function of distance, radial position and propeller loading). As mentioned in the introduction, a completely numerical lifting-line method has been developed to calculate circumferential and longitudinal interference factors. The lifting-line model is similar to that of Lerbs (1952), but it has been changed from a continuous formulation into a discrete one by employing vortex lines composed by constant length parts. So, induced velocities are not evaluated by the well-known analytical formulations, but by simply applying the Biot-Savart law to the discrete bound vortex elements and to the free trailing vortices.

2.2 Fully Numerical Approach

The fully numerical design approach for contra-rotating propeller is based on the original idea of Coney (1989) for the definition of optimum radial circulation distribution for lightly and moderately loaded propeller in non-uniform inflow. Traditional lifting-line approaches are, as presented above, mainly based on Betz criteria for the minimum energy loss on the flow downstream of the propeller. The satisfaction of these conditions is realized by an optimum circulation distribution, generally defined
as a sinus series over the blade span. In the fully
numerical design approach, this continuous distribution of
vorticity along the lifting line representing each constant
angular spaced blade of the propeller is discretized with a
lattice of vortex elements of constant strength. The
continuous trailing vortex sheet that represent the blade
tailing wake is therefore replaced by a set \( M \) of vortex
horseshoes of intensity \( \Gamma(m) \), each composed by two
helical trailing vortices and a bound vortex segment on
the propeller lifting line, as in Figure 2. Also, the
propeller hub effect can be included by means of image
vortices based on the well-known principle that a pair of
two-dimensional vortices of equal and opposite strength,
located on the same line, induce no net radial velocity on
a circle of radius \( r_h \):

\[
r_i = \frac{r_h^2}{r}
\]

where \( r \) is the radius of the outer vortices, \( r_i \) the radius
of its image and \( r_h \) the radius of the hub cylinder. The same
result approximately holds in the case of three
dimensional helical vortices, provided that their pitch is
sufficiently high.

This system of discrete vortex segments, bound to the
lifting line or part of each helical vortex line trailed in
the wake (Figure 2) induces axial and tangential velocity
components on each control points of the lifting line.
These self-induced velocities are computed applying the
Biot-Savart law, in such a way:

\[
\begin{align*}
u_a(r_i) &= u_a(i) = \sum_{m=1}^{M+M_h} A_{im} \Gamma(m) \\
u_t(r_i) &= u_t(i) = \sum_{m=1}^{M+M_h} T_{im} \Gamma(m)
\end{align*}
\]

where \( A_{im} \) and \( T_{im} \) are, as usual, the axial and tangential
velocity influence coefficients of a unit horseshoe vortex
placed at the \( m \)-radial position on the \( i \)-control point on
the lifting line, and \( M \) and \( M_h \) are the total number of
horseshoe vortices on the blade and on the image hub.
With this discrete model, the hydrodynamic thrust and
torque characteristics of the propeller can be computed by
adding the contribution of each discrete vortex on the
line. In fact, under the assumption of pure potential and
inviscid flow:

\[
\begin{align*}
T &= \rho Z \int_{r_h}^{R} V_{tot} \cos \beta_t \, dr \\
Q &= \rho Z \int_{r_h}^{R} V_{tot} \sin \beta_t \, dr 
\end{align*}
\]

where \( V_{tot} \cos \beta_t \) is simply the total tangential velocity
acting at the lifting line (inflow \( V_t \) plus self induced \( u_t \)
plus rotational tangential velocity \( \omega r \)), \( V_{tot} \sin \beta_t \) is the
axial velocity (inflow \( V_a \) plus self induced axial velocity
\( u_a \)), and \( \beta_t \) is the local angle of attack. In discrete form,
equation (3) can be expressed as:

\[
T = \rho Z \sum_{m=1}^{M} [V_t(m) + \omega r + u_t(m)] \Delta r \Gamma(m)
\]

\[
Q = \rho Z \sum_{m=1}^{M} [V_a(m) + u_a(m)] \Delta r \Gamma'(m) r_m
\]

A variational approach provides a general procedure to
identify a set of discrete circulation values \( \Gamma(m) \) (i.e.,
the radial circulation distribution for each propeller blade)
such that the torque (as computed in Equation (3)) is
minimized subjected to the constraint that the thrust must
satisfy the prescribed value \( T_R \).

Introducing the additional unknown represented by the
Lagrange multiplier \( \lambda \), the problem can be solved in terms
of an auxiliary function \( H = Q + \lambda (T - T_R) \) requiring
that its partial derivatives are zeros:

\[
\frac{\partial H}{\partial \Gamma(i)} = 0 \quad \text{for} \quad i = 1..M
\]

\[
\frac{\partial H}{\partial \lambda} = 0
\]

Carrying out the partial derivatives, Equation (9) leads to
a nonlinear system of equations for the vortex strengths
and for the Lagrange multiplier. The iterative solution of
the nonlinear system is obtained by the linearization
proposed by Convey (1989) in order to achieve the optimal
circulation distribution. This formulation can be further
improved to design moderately loaded propeller and to
include viscous effects. The initial horseshoe vortices
that represent the wake, frozen during the solution of
Equation (9), can be aligned with the velocities induced
by the actual distribution of circulation and the solution
iterated again until convergence of the wake shape (or of
the induced velocities themselves). A viscous thrust
reduction, as a force acting on the direction parallel to the
total velocity and thus as a function of the self-induced

---

**Figure 2: Discretized lifting line and reference system.**

Introducing the additional unknown represented by the
Lagrange multiplier \( \lambda \), the problem can be solved in terms
of an auxiliary function \( H = Q + \lambda (T - T_R) \) requiring
that its partial derivatives are zeros:

\[
\frac{\partial H}{\partial \Gamma(i)} = 0 \quad \text{for} \quad i = 1..M
\]

\[
\frac{\partial H}{\partial \lambda} = 0
\]

Carrying out the partial derivatives, Equation (9) leads to
a nonlinear system of equations for the vortex strengths
and for the Lagrange multiplier. The iterative solution of
the nonlinear system is obtained by the linearization
proposed by Convey (1989) in order to achieve the optimal
circulation distribution. This formulation can be further
improved to design moderately loaded propeller and to
include viscous effects. The initial horseshoe vortices
that represent the wake, frozen during the solution of
Equation (9), can be aligned with the velocities induced
by the actual distribution of circulation and the solution
iterated again until convergence of the wake shape (or of
the induced velocities themselves). A viscous thrust
reduction, as a force acting on the direction parallel to the
total velocity and thus as a function of the self-induced
velocities themselves, can be added to the auxiliary function $H$, and a further iterative procedure, each time the chord distribution of the propeller has been determined, can be set. In total, for the design of a single propeller, the devised procedure works with:

- An inner-iterative approach for the determination of the optimal circulation distribution,
- A second-level iterative approach to include the viscous drag on the optimal circulation distribution,
- A third-level iterative approach to include the wake alignment and the moderately loaded case.

Contra-rotating propellers requires an equivalent approach, defining an auxiliary function as a combination of thrust and torque of the two propellers and carrying out differentiations with respect to the vortex strengths distributed on the two contra-rotating lifting lines.

However, the foregoing design procedure for the single propeller can be successfully extended to the contra-rotating case. Each propeller can be designed as single, wake adapted propeller, whose inflow induced by the other propeller is computed in an circumferentially average way. A further external and iterative scheme drives the design until a satisfactory convergence on the required total thrust is achieved.

### 2.3 Geometry Definition

After defining the blade circulation and hydrodynamic pitch distribution, the design procedure proceeds to determine the blade geometry in terms of chord length, thicknesses, pitch and camber, which ensure the requested section lift coefficient while also ensuring cavitation and strength constraints. As already mentioned, this design stage follows the same procedure for both the foregoing methods. There are several approaches when solving the foregoing problem, but the authors follow the works by Connolly (1961) for the calculation of blade stresses and the method proposed by Grossi (1980) for cavitation issues. The last procedure is based upon an earlier work by Castagneto et al (1968) where minimum pressure coefficients on a given blade section with standard (NACA) shapes are semi-empirically derived. The other semi-empirical simplified blade model proposed by Connolly considers radial stresses $\sigma_r$ due to bending moment and radial stresses $\sigma_c$ due to centrifugal forces: the former are determined by the developed thrust and absorbed torque, while the latter is determined by the propeller revolutions and blade thickness distribution $t(r)$. Starting with an initial guess for $t(r)$, the program computes the required blade sectional modulus (given the maximum admissible stress value), and proceeds to determine chord length distribution $c(r)$ by making use of a simplified cavitation inception rule where minimum pressure coefficient $C_{p,\text{min}}$ on each blade section has to be higher than the cavitation number $\sigma_0^c$ times a safety factor $K_p$ used to ensure a certain design margin (usually $K_p < 1$):

$$K_p\sigma_0^c = C_{p,\text{min}}$$  \hspace{1cm} (10)

In order to compute $C_{p,\text{min}}$, the authors follow Castagneto and Maioì’s work, where thin profile theory for standard propeller foils and mean lines is corrected by empirical results. Given the blade sectional modulus and $c(r)$, it is possible to derive a new distribution for $t(r)$ and the foregoing procedure is repeated until satisfactory convergence is achieved on the parameters involved. Then the program proceeds to the calculation of the pitch angle and camber distribution according to the required blade circulation. At the last stage, lifting surface corrections are applied to the foregoing distributions according to the method devised by Van Oossanen (1968) and revised by the authors to account for the contra-rotating case.

### 3 ANALISYS PROCEDURE

#### 3.1 Lifting Line and Panel Method

The first analysis tools applied to the resulting design geometries are two potential flow codes developed by the authors and described in detail in previous works (Brizzolara et al. 2008, Grassi & Brizzolara 2007, Gaggero & Brizzolara 2009). Assuming the flow as inviscid, incompressible and irrotational, the general continuity and momentum equations lead to a Laplace problem for a scalar function $\phi$:

$$\nabla^2 \phi(x) = 0$$  \hspace{1cm} (11)

The choice of the harmonic functions that satisfy Equation (11) determines the solution. The first solver is a lifting surface based code: the blade geometry is approximated by its mean surface and the continuum vortex sheet lying onto is discretized by means of a certain number of vortical rings. The second one is a panel method in which both the blades and the hub are discretized with hyperboloidal panels carrying constant strength sources and dipoles (mathematically equivalent to vortex rings). A certain number of boundary conditions, depending on the nature of the problem, are thus applied in order to satisfy Kutta condition at blade trailing edge, to satisfy the kinematic condition on solid boundaries and to satisfy the force free condition for the trailing vortical wake. Since the physics of a contra-rotating propeller set is a strongly unsteady phenomenon even in open water configuration, it should be treated in time domain. In order to simplify the computational effort, a steady iterative approach has been followed in the present work: the interaction of the fore and aft propeller is intended in the sense that each propeller is solved as a single propeller in a wake-adapted condition where the inflow velocity is calculated by solving the other propeller, calculating the velocity field in a transversal plane axially located in correspondence of the other propeller and taking the mean value of the axial, tangential and radial component in the circumferential direction. The velocity field downstream of the fore
propeller and upstream of the aft one is expressed in cylindrical coordinates as a function of the radial position:

\[ V(r) = V_a(r) + V_r(r) + V_t(r) \]  

where every component is given by:

\[ V_a(r) = \frac{\int_{\theta_1}^{\theta_2} V_a(r, \theta) d\theta}{\theta_2 - \theta_1} \]

\[ V_r(r) = \frac{\int_{\theta_1}^{\theta_2} V_r(r, \theta) d\theta}{\theta_2 - \theta_1} \]

\[ V_t(r) = \frac{\int_{\theta_1}^{\theta_2} V_t(r, \theta) d\theta}{\theta_2 - \theta_1} \]

3.2 RANS Solver

The analysis of contra-rotating propeller performances has been finally carried out with the adoption of StarCCM+, a finite volume commercial RANS solver. Two level of approximation for the solution of the unsteady problem are possible. In the first case, denoted as “quasi steady”, the interaction between the front and the rear propeller has been considered steady, as the relative position between blades of the two propellers never changes. To have a better insight into the unsteady behavior of the contra-rotating flow within the quasi steady approach, a set of different relative angle positions has been computed. Proposed results are the average of propellers characteristics at different relative positions.

![Figure 3: Quasi Steady domain decomposition at a given relative angular position.](image)

This is obviously true for co-rotating propellers, while contra-rotating propellers should be instead properly designed to exploit the change in the relative position. However, this approximation, which is almost the same as adopted for the iterative analysis carried with lifting surface and panel methods, allows to strongly reduce computational times and mesh complexity. With this choice, in fact, it is possible to model only one blade per propeller by applying proper periodic conditions at the boundaries and to use a moving reference frame solver that speeds up the solution time. This means, nevertheless, that the rear propeller blade is forced to operate in a fixed, steady position with respect to the inflow wake produced by the front propeller blade, ignoring the time/spatial inflow variations due to the entire set of front blades rotating in the opposite sense. Accurate solutions have been obtained with this approach with about 700k cells for the entire domain, front plus rear region, as in Figure 3, with computational time affordable with a modern personal computer.

![Figure 4: Fully Unsteady domain decomposition](image)

4 RESULTS

Design procedures (Morgan-Lerbs and fully numerical) and analysis approaches (lifting line, panel method and RANS) have been compared in the case of a set of contra-rotating propellers whose requirements are to produce a total thrust of about 30kN (30154N) at constant revolution of 1040 rpm, corresponding to an advance speed of about 36kn as per Table 1.

### Table 1: Design parameters for the contra-rotating propellers

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Front Propeller</th>
<th>Rear Propeller</th>
</tr>
</thead>
<tbody>
<tr>
<td>( V_a ) [kn]</td>
<td>36</td>
<td>36</td>
</tr>
<tr>
<td>( \omega ) [rpm]</td>
<td>1040</td>
<td>1040</td>
</tr>
<tr>
<td>( Z )</td>
<td>4</td>
<td>4</td>
</tr>
<tr>
<td>( D ) [m]</td>
<td>0.765</td>
<td>0.744</td>
</tr>
<tr>
<td>( r_i/R )</td>
<td>0.25</td>
<td>0.257</td>
</tr>
<tr>
<td>Front-Rear distance</td>
<td>0.27 m</td>
<td></td>
</tr>
</tbody>
</table>

Three different design solutions are obtained and discussed in the following paragraphs: in the first case, a 50%-50% thrust distribution between fore and aft propeller is imposed in order to compare the fully numerical design with the Morgan-Lerbs, where a balanced thrust/torque distribution is inherent in the theoretical approach. The fully numerical procedure has
been successively adopted for the design of unbalanced contra-rotating propellers, with a distribution of thrust equal to 60%-40% and 40%-60% thrust distribution to explore possible advantages in terms of overall open water efficiency.

All the proposed designs have been obtained accounting for the aforementioned semi-empirical model for viscous drag corrections and the wake alignment procedure, also included in the analysis tools. The four-bladed solution for both the front and the rear propeller has been selected as a good compromise between efficiency and cavitation risk reduction.

4.1 50-50 Contra-rotating propeller

The balanced design has been defined through the fully numerical program for two different configurations, including the hub effect via the image model and without the hub, in order to consider design conditions closer to the Morgan-Lerbs hypothesis. Figures 5 and 6 compare front and rear propeller circulation distributions; the resulting self-induced velocities are compared in Figures 7 and 8. Significant differences can be noted. Front propeller circulation computed with the fully numerical approach has a different shape: the maximum value is higher and located in a different radial position with respect to the Morgan-Lerbs distribution.

![Figure 5: Front propeller circulation distribution.](image)

![Figure 7: Self-induced velocities (design). Front propeller.](image)

![Figure 6: Rear propeller circulation distribution.](image)

![Figure 8: Self-induced velocities (design). Rear propeller.](image)

The presence of the image vortex, as expected, loads the distribution near the hub, but also, the design, performed without the hub effect, has a value different from zero (partially due to the numerical discretization of the problem), whereas Morgan-Lerbs circulation (intended as a sine series summation) is null at the hub radius due to the mathematical treatment of the problem. Circulation distributions for the rear propeller are similar, in particular, for the modified Morgan-Lerbs method and the fully numerical without hub (Figure 6). The presence of the hub, also in this case, significantly changes the load distribution. For the sake of completeness, the circulation distributions were also calculated (with the fully numerical algorithm); neglecting the viscous thrust, reduction and the wake alignment are compared with fully numerical and the Morgan-Lerbs approach. Absence of viscous correction and of wake alignment modifies the induced velocities and, consequently, the circulation distributions that are slightly unloaded. Obviously, if viscosity is neglected, the propeller delivered thrust is exactly the required one. If, instead, viscous thrust corrections are accounted, the propeller has to deliver a greater thrust (and, thus, to produce a more loaded circulation) to balance the viscous losses. Finally, it has to be noted that Morgan-Lerbs Equivalent Propeller approach produces a single averaged circulation distribution for both the propellers, whereas the fully numerical design procedure computes different optimum circulation distributions as a function of the different inflow condition due to the mutually induced velocities. This effect is shown in Figure 9 in terms of axial and tangential components.
In terms of self-induced velocities, the difference between the two codes correctly follows the noted difference between bound circulation distributions. Self-induced velocities computed by the two codes for the fore propeller (Figure 7) evidence the major differences between the design methods, especially with regard to the tangential component at the inner radial positions. Since circulation distributions for the rear propeller are very close, the related self-induced velocities are more similar (Figure 8) and only the addition of the hub produces a noticeable influence (Figure 9). A further comparison of induced velocities is presented in Figures 10 and 11, in which mean induced velocities on the rear propeller plane, computed with panel and lifting surface methods, are compared with induced velocities predicted by the design codes. Attention is focused on the numerically designed propellers, including hub and on those obtained with the modified Morgan-Lerbs method; induced velocities computed by the analysis codes, namely panel and lifting surface codes are compared, respectively, with induced velocities predicted by the fully numerical approach and by the Morgan-Lerbs procedure. This comparison is useful in light of the iterative nature of the design and of the analysis codes that treat each propeller of the contra-rotating as a single propeller designed or operating in the averaged inflow induced by the other one. Induced velocities by design procedure and respective analysis codes are quite close. The hub, when considered (panel method and fully numerical design approach), significantly alters the mean distribution of induced velocity on the rear propeller plane. For the numerical designed propeller, as in Figure 10, the presence of the hub during the design phase increases axial and tangential induced velocities at the inner radial position. Panel method captures well the presence of the hub predicting finite value of axial and tangential velocities, close to those computed with the numerical design procedure. Some differences are still present and can be attributed to the exact modeling and to the panel method of the mathematical singularities (sources) that represent thickness, which is instead neglected (hub) or approximated (profile thickness) in the numerical lifting-line design approach and in the lifting surface analysis code.

Velocities predicted by the lifting surface code, even if close to the design values near the tip, have a different behavior at the hub, especially for what regards the tangential component.
margins between the two design methods, are quite close, with major differences at the hub. Only at the tip, for radial positions greater than $r/R = 0.6$, where circulations and velocities are close, pitch and camber for the two designs are comparable; also if peculiarities related to the zero value for the circulation of the Morgan-Lerbs design are still clear. At the inner radial positions, where hub loading influences circulation and velocities, differences are noticeable, especially for the front propeller pitch, for which the gap in velocities between the Morgan-Lerbs and the numerical design with hub is greater. The hub loading effect is, finally, clear comparing camber distributions. Higher values of circulation/velocity, computed by the fully numerical design that includes the hub effect, are responsible of greater values of local lift coefficient ($C_L \cdot chord$) that, in turn, determines how much load is produced by angle of attack (pitch) and by camber. The higher the local lift coefficient is, the higher the local maximum camber has to be.

Despite these differences noted on local load and pitch and camber distributions, the overall performances, computed with several analysis approaches, are quite close. The three different designs have total thrust values, as presented in Figure 14, close to the prescribed one in whichever analysis method is employed. Propellers designed by the fully numerical procedure, including the hub when analyzed with the panel method that also includes the hub effect, gives the 99% of the prescribed design thrust (30154 N). RANS Quasi Steady (QS) and Fully Unsteady (FU) computations, being able to include hub effects and to theoretically reproduce the most realistic flow, are believed to be more accurate than the panel method; these methods result in 101% and 103%, respectively, of the prescribed design thrust. Hence, propellers designed with the fully numerical approach are characterized by slightly higher values of thrust with respect to the design requirements.

The presence of the hub increases, as presented in Figures 12 and 13, the local load, determining very high values of local lift and, thus, of local maximum camber. Such high values of camber are, in general, out of a common design practice and tend to produce back/face inversion on the profile pressure distribution. Neglecting the hub results in camber distributions closer to the Morgan-Lerbs design and in balanced higher values for the pitch.
overestimate propeller performances, result only in 97% of the prescribed design thrust; on the other hand, the lifting surface code results in 100% of the input design thrust, suggesting the general tendency of lifting surface to overestimate the developed forces. The predicted thrust using RANS QS and FU shows no significant difference when compared with the prescribed thrust, thus confirming to be slightly above the panel method. Lifting surface results, on its own, seems in general to be in line with RANS computation with a slight tendency to overestimate the performances.

When comparing the three designed geometries in terms of delivered thrust, it is evident that the numerical designs tend to overestimate whereas the Morgan-Lerbs approach appears very close to the original design input value. Torque computations confirm the trend highlighted above for thrust. In absence of experimental measurements, RANS Quasi Steady and Fully Unsteady values of torque, that are particularly close each other for all the three designs, could be considered as a reference point for validation. It is clear, for torque more than for thrust, that lifting surface over-predict forces a bit (about 7.5% in the average for the three designs), while panel method,
although underestimating results (about 3.5% in the average), is closer to the reference RANS values. Moreover, the cavitation free design constraint (required for both the design approaches) has been confirmed by all the available analysis tools.

Finally, Figures 16-18 and 19-20 show the velocity distribution and the vorticity magnitude computed by the RANS in fully unsteady conditions on a longitudinal section and on plane 0.5D downstream the rear propeller for both the designed sets.

4.2 40-60 & 60-40 Contra-rotating propellers

Unbalanced distributions of thrust between the front and the rear propeller are possible only through the fully numerical design approach. Two alternative designs have been performed: the first with 60% of the thrust delivered by the front propeller (40% by the rear), and the second with 60% of the thrust delivered by the rear propeller (40% by the front). Overall performances, computed by lifting surface, panel method and RANS quasi steady approaches (as demonstrated in the previous computations differences between quasi steady and fully unsteady computations are negligible) are compared, in Figures 21 and 22, to the balanced propeller design.

Table 2: Unbalanced propeller comparison – Quasy Steady computations.

<table>
<thead>
<tr>
<th>Total $T$ [N]</th>
<th>Total $Q$ [Nm]</th>
<th>Efficiency</th>
</tr>
</thead>
<tbody>
<tr>
<td>50-50</td>
<td>30566</td>
<td>6411</td>
</tr>
<tr>
<td>60-40</td>
<td>31189</td>
<td>6556</td>
</tr>
<tr>
<td>40-60</td>
<td>30517</td>
<td>6405</td>
</tr>
</tbody>
</table>

Also in this case of unbalanced designs, all the analysis tools show very good agreement at the design point. RANS computations are about 1.5% above the thrust design input value, while panel method is slightly below (-1% in the average). The lifting surface code meets the target input very well also. With regard to torque values, assuming RANS computations as a reference line, lifting surface computations confirm the over-prediction tendency (6-7%), whereas the panel method is generally closer (-3%).

Figure 22: Comparison of computed torque.

As showed in Table 2, when the open water efficiency is considered, the three design solutions are very close.

4 CONCLUSIONS

Two different computational methods for the design of contra-rotating propellers, based on lifting-line theory, have been presented in the paper: one fully numerical, the other based on a revisited Morgan-Lerbs theory. Three different numerical tools have been used to analyze the resulting blade geometries. Global values such as thrust and torque, as well as local parameters as the induced velocities and bound circulation have been compared. The analysis first demonstrates that both the design methods produce geometries that satisfy input constraints and requirements. Moreover, it is also clear that all the analysis tools employed are able to deal with the contra-rotating propeller problem with sufficient accuracy for engineering purposes. Fully Unsteady RANS computations are the most realistic way to reproduce flow dynamics around a contra rotating set, while Quasi Steady RANS and Potential solutions are able, as well, to give a practical and realistic prediction of propellers (circumferentially) averaged forces.

However, some crucial differences in the design procedure can be highlighted. The inclusion of the hub in the fully numerical design method produces major differences in the load distribution and consequently geometry parameters, such as pitch and camber. The higher values of local lift coefficient associated to the hub wall effect for the inner radial sections are responsible of the unusually high values of maximum profile camber in these positions.

Lifting surface corrections (Van Oossanen 1968), developed and validated for the original no hub design approach, not including in turn this effect, enhance this difference, thus requiring hub unloading correcting functions, usually derived from the experience.

On the other hand, the fully numerical design procedure, together with the panel method, permit to investigate unusual configurations that would otherwise be impossible to obtain by the traditional Morgan-Lerbs method. Unbalanced propellers have been successfully designed and numerically validated, which can lead to the design of contra-rotating set characterized by different
number of blades and different rate of revolutions between the front and the rear propeller. The possibility to have different number of blades between the fore and the aft propeller could help in avoiding the potential risk of resonance subsequent to the choice of an equal number of blades.

REFERENCES


