Interactive Program System for the Design/Analysis of Marine Propulsors

Vladimir I. Krasilnikov, PhD, Research Scientist,
Aage Berg, MSc, Senior Research Engineer
MARINTEK – Norwegian Marine Technology Research Institute, Trondheim, Norway

Alexander S. Achkinadze, D.Tech.Sc., Prof.,
Ivan E. Stepanov, PhD
State Marine Technical University, St Petersburg, Russia

ABSTRACT

This paper describes the program system intended for the practical designing of marine propulsors. The current program version includes propeller design program based on non-linear lifting surface theory, analysis program based on original boundary element method and visual tools for the input/output data control and edition, drawing and creation of 3D propeller model for milling machines. The main details of numerical implementation and some recent validation results are presented and the examples of practical designing are discussed.

1 INTRODUCTION

Nowadays the computer based tools have become an inseparable part of the arsenal of propeller designer. The more and more extensive use of numerical algorithms is caused not only by the considerations of time and cost saving, but also by the wide possibilities, which such the algorithms demonstrate in the field of systematic study and investigation of fine effects that can be hardly treated only by the experimental methods. For example, the problem of propeller trailing edge modeling in model scale appeared with the advent of NURBS surface representation (Michael and Jessup 2001), or effect of the leading edge form on pressure side cavitation (Achkinadze et.al. 2002). In some cases the improvement of the existing propeller design by keeping the main propeller elements at “desirable” pressure distribution on the blade surface is a point of interest (Tan 2001). In last years many propeller manufacturers earlier dealing with conventional propulsion systems met the new challenges thrown down by the wide application of POD arrangements (Hämäläinen and Heerd 2001), (Friesch 2001). Design of podded propellers is principally different from traditional single propeller design and requires accounting for the interaction between the propulsor components. In fact, there are no ready “diagram” solutions for podded propellers, and the designer faces new research problems almost in every project.

In the propeller design problem the lifting line/lifting surface technique still remains the main well-verified and reliable tool. It is, as a rule, the part of any integrated algorithm intended for the design of multi-component propulsors, which is responsible for definition of blade pitch and camber distribution at given conditions. The interaction between propeller and pod unit in the design methods is taken into account in the potential formulation via velocity field iterations (Achkinadze and Krasilnikov 2000), or, in more advance algorithms, via coupling with the axisymmetrical viscous solvers for the pod body (Kerwin et.al. 1997), (Hsin et.al. 2002).

At the same time the analysis methods reveal much greater variety. Along with the purely potential lifting surface (Yang et.al. 1992), (Hughes and Kinnas 1993), (Chun et.al. 2001) and boundary element (Ghassemi and Allievi 1999) methods the coupled lifting-surface/RANS (Kerwin et.al. 1997), (Warren et.al. 2000) and
BEM/Euler-solver (Kinnas et al. 2001) techniques are used. The direct RANS simulations of podded arrangements are also known (Sanches-Caja et al. 1999), although such the works are rather research studies than standard procedures.

The interactive program system for the design/analysis of multi-component marine propulsors described in the present paper is the result of long-term cooperation between MARINTEK, Trondheim, Norway and State Marine Technical University of St Petersburg, Russia. It allows numerical simulation of conventional open propellers (fixed and controllable pitch), propeller/rudder systems, shaft and pod two-staged arrangements. The propeller design algorithm is based on the non-linear lifting surface theory and Generalized Linear Model (GLM) of the trailing vortex wake (Achkinadze and Krasilnikov 1997), (Achkinadze and Krasilnikov 1999), (Achkinadze et al. 2000). It handles two main design variants: 1) design of propeller with determined (prescribed) characteristics, when all the elements of propeller are given and only pitch and camber distributions are the subjects for the design calculation; 2) optimization design of wake-adapted propeller, when the searched optimum circulation distribution along the radius allows the maximum propulsor efficiency at specified conditions. In the latter case the special Generalized Optimum Condition (GOC) takes into account the tangential component of velocity field, i.e. flow swirl, that is important when designing two-staged propulsors.

The analysis procedure for propeller and rudder/strut is the original boundary element method (BEM) developed by the authors (Achkinadze and Krasilnikov 2001), (Achkinadze et al. 2001), (Achkinadze and Krasilnikov 2003), (Achkinadze et al. 2003). This is the velocity based source BEM, which uses the algorithm of Modified Trailing Edge for the direct satisfaction of the Kutta-Joukowski condition when defining unknown doublet strength and for the approximate accounting for viscous effects on circulation.

In the both design and analysis algorithms the same vortex model of propeller wake (GLM) is used, and the interaction between the propulsor components is taken into account via mutually induced velocity fields (method of velocity field iterations). The trailing vortices from propeller blades and strut are allowed to freely penetrate through the adjacent bodies as if the latter would be absent and they do not experience any deformation, while the kinematic boundary condition is strictly satisfied on the solid boundaries (hypothesis on penetrability). The propeller-induced velocity fields are always considered as circumferentially averaged, while the pre-set hull wake and velocity fields induced by the pod and strut in the unsteady analysis are, of course, spatially variable.

The design of a multi-staged propulsor is an iterative process, on each step of which the user should check the different elements from the principal arrangement of the stages up to blade surface geometrical characteristics, pressure distribution and cavitation performance. Therefore, in the developed software special attention has been paid to convenient user-friendly interface and possibilities of visualization. The current visual tools allow clear representation of pressure distribution and cavitation domains on propeller blades, rudder/strut) and pod surfaces and velocity fields in the propeller planes. The possibilities of manual change of blade geometry and data transfer between design, analysis, drawing and surface generation programs allow the user to organize the design process as a dialog with the program.

The developed design and analysis methods have passed through detailed numerical/experimental verification, and some of the examples are considered in the paper, as well as the examples of use of the program system in design practice.

2 PROPELLER DESIGN ALGORITHM

The propeller design problem, i.e. definition of the blade pitch and camber distributions along the radius, is a kernel of the design algorithm for multi-component propulsor. In dependence upon the given conditions and particular requirements, different levels of optimization are possible. In (Achkinadze et al. 2000) the following five levels of optimization are distinguished beginning from the case of determined design when all the propeller element including spanwise circulation distribution are given (level 1) through the optimization of spanwise circulation (level 2), optimization of chord length and thickness distributions at required strength and cavitation margins (level 3), the same problem, but with simultaneous optimization of circulation (level 4) and up to the optimization of propeller diameter and rate of revolution at simultaneous solution of the level 4 problem (level
5). The current version of the program system handles the first two cases as the most frequent in design practice, although the algorithm for the rest three problems have also been built and described in (Achkinadze et.al. 2000). The calculation of propeller is performed in pre-set circumferentially averaged velocity field, which in the case of single propeller is the radially variable hull wake specified in the propeller plane, while in the case of multi-component propulsor this field includes velocities induced by all the components. In the latter case the flow field onset propeller is considered as radially and axially variable. These tasks are solved at the first step within the frameworks of the lifting line theory and here the choice of a proper vortex model is of high importance.

2.1 Vortex wake model and optimum condition

In spite of that in a great number of “ordinary” cases the different vortex models allow successful design with reasonably close results (ITTC 1987), the reliable mathematical model has to remain valid in all the possible situations and has to be suitable for correct solution of the optimization problem. The well-known case with the paradoxical results predicted by the Lerbs model at overloaded outer blade sections (Loukakis and Politis 1981), (Achkinadze and Mishkevich 1986) shows how carefully the researcher should select the non-linear models for the numerical procedure. In fact, there are two possible ways of treating this problem: first, development of the quantitatively non-linear model, for example, semi-empirical with N number of external parameters (Greeley and Kerwin 1982) or a model with full wake alignment (Kinnas and Pyo 1997), and, second, use of the simplified linear theory, which is free from the mentioned drawbacks. In authors’ opinion in the propeller design problem the second way is more preferable since at the very first step of design process the choice of the external parameters can be problematic, especially in the case of wake-adapted propeller or multi-staged propulsor. At the same time, the use of the wake alignment models belongs rather to the analysis problem, and does not seem very efficient in the design where the blade geometry is a subject for calculation.

The developed Generalized Linear Model (GLM) (Achkinadze and Krasilnikov 1997) demonstrates good computational performance at simple fixed wake geometry. It assumes the constant pitch of the free vortex surfaces (they are the regular helicoids), which is taken as equal to the hydrodynamic pitch of the equivalent isolated propeller designed using the Prandtl’s optimum condition. It is important to point out that in this model the pitch of free vortices does not depend upon the local flow velocities (that conforms the linearity of the approach), but it is defined by the integral performance of propeller (design $K_T$ or $K_Q$) and averaged parameters of the inflow (axial wake coefficient and equivalent swirl).

Another important feature consists in that only within the frameworks of GLM the Generalized Optimum Condition (GOC) is proved to be valid. The GOC was first published by Achkinadze in 1985 and it generalizes the other well-known optimum conditions such as Betz (1919), Prandtl (1919), Slutski (1940), Burill (1944), Lerbs (1952) and Yim (1978) conditions (Achkinadze and Krasilnikov 1997). Below the asymptotic form of GOC, which has been used in the current numerical algorithm, is given

$$\frac{w_{n1}}{\pi n R \cos \beta_{FVS}} = c^* \frac{J_Y}{\pi} \left( t + \frac{U_X}{r} \right) - \frac{J_Y}{r/R} \frac{\pi}{U_\theta},$$  

where $J$ is the advance coefficient based on ship speed, $R$ is the propeller radius, $r$ is the radius of the blade section, $n$ is the propeller rate of revolution, $w_{n1}$ is the propeller-induced velocity at the lifting line normal to the free vortex surface, $\beta_{FVS}$ is the pitch angle of free vortices, $U_X$ and $U_\theta$ are respectively axial and tangential components of the inflow related to ship speed (here $U_X$ is positive downstream, $U_\theta$ is positive to propeller rotation), $t$ is the thrust deduction factor and $c^*$ is the constant defined from the isoperimetric condition of given thrust or torque. The performed calculations have shown that the difference in propeller efficiency between full and asymptotic forms of GOC usually does not exceed 0.1% even at high propeller loading. At the same time the gain due to accounting for the tangential velocity component (last item in the right-hand part of (1)) may achieve 2% compared to the Prandtl’s optimum condition in highly swirled flows. The
direct comparison between GOC and the procedure of numerical optimization (Kerwin et al. 1986) performed in (Achkinadze and Krasilnikov 1997) has shown very close optimum circulation distributions predicted by both methods in the case of the open propeller in uniform flow. However, it is important to note, that GOC provides the solution, which is “very close” to the global optimum in all possible cases.

2.2 Iterative geometry alignment procedure

The ordinates of the blade mean surface, which then are recalculated in the standard pitch and camber distributions, are derived from the so-called lifting-surface design equation (Achkinadze and Krasilnikov 2000):

\[
\frac{\partial \eta'}{\partial \xi'} = \left[ V_{r\xi} - V_{r\eta} \cdot \left( \frac{\partial z_c^0}{\partial y^0} \right) \right] / V_{r \xi},
\]

(2)

where \(\xi'\) is the chordwise coordinate counted along the section nose-tail line, \(\eta'\) is the ordinate of the mean surface, \(V_{r\xi}, V_{r\eta}\) and \(V_{r\nu}\) are the tangential to chord, normal to chord and radial components of the total relative velocity, \(\frac{\partial z_c^0}{\partial y^0}\) is the partial derivative, which characterizes the amount of blade rake and is calculated in the local coordinate system fixed on given point on the blade chord. Obviously, the ordinates of the mean surface (\(z_c^0\) in the local coordinate system) are not known before the solution. Therefore, the written equation has to be solved in iteration. This iterative process forms the so-called geometry alignment procedure. At the first step the equation (2) is applied in its linearized form, which follows from the main form at \(\frac{\partial z_c^0}{\partial y^0} = 0\). This step corresponds to the pure GLM. In the linearized lifting surface formulation we omit the radial component of the relative velocity. However, in the case of moderately skewed and raked blades already this first step gives sufficient accuracy that has been confirmed by the many comparisons (Achkinadze and Krasilnikov 1997). When the designed blade has significant amount of skew and rake, and when the propeller is designed for the strong radial crossflow (as it takes place for the blade row installed on highly conical centerbody) the full form of (2) allows higher precision in prediction of pitch and camber values. The numerical tests have shown relatively fast convergence of the geometry alignment procedure. In most of the studied cases stable values of pitch and camber were obtained after 3-5 iterations.

2.3 Design of multi-component propulsor

The interaction between the components of designed propulsor is taken into account via velocity field iterations. Since the design calculation is always performed in circumferentially averaged flow the fast and efficient procedure for computation of propeller-induced velocity field can be built on the base of the infinite-bladed propeller theory by Hough and Ordway (Hough and Ordway 1965). Along with the adopted hypothesis about penetrability of the bodies for the trailing vortices this approach allows us to overcome the numerical singularities unavoidable when performing direct integration on the free vortex surfaces. The identities for the axial and radial mean propeller-induced velocity components are obtained as 0-harmonics from the general expressions for the complex Fourier coefficients through the Legendre functions of the 2nd kind and half-integer order as follows (Achkinadze et al. 2003):

\[
\overline{W}_s = \frac{1}{2\pi} \int_{r_n}^{R} \gamma_{FW}^R (r) K_1(x_1, r_1, r) dr,
\]

(3)

\[
\overline{W}_r = -\frac{1}{2\pi} \int_{r_n}^{R} \gamma_{FW}^R (r) r^1 Q_{1/2} (\omega_1) dr,
\]

(4)

where \(K_1\) and \(Q_{1/2}\) are calculated analytically through the Legendre special functions and \(\gamma_{FW}^R (r)\) is the strength of the ring vortices at the given radius \(r\), which is connected with the circulation derivative.
\[
\gamma_{FW}^R (r) = -Z \frac{d\Gamma}{dr} (r) dr \frac{1}{2 \pi r \tan \beta_{FVS}}.
\]  

It has to be noted that the identities (3)-(5) are different from ones originally obtained by Hough and Ordway because the pitch angle of the free vortices \( \beta_{FVS} \) is taken according to GLM as it has been described above. The averaged tangential propeller-induced velocity in the propeller wake is defined by the well-known formula through the circulation

\[
\overline{W}_\theta = \frac{Z \Gamma (r_1)}{2 \pi r_1}.
\]

Thus, the only procedure of numerical integration is necessary for computation of the integral along the radius in (3) and (4), and the singularity in behaviour of \( d\Gamma / dr \) on the end of the interval can be easily overcome by the application of the special quadratures for singular integrals (Krasilnikov 1997).

In the design algorithm the blade forces are calculated using the empirical corrections accounting for the profile drag and, therefore, a reduced circulation value is obtained due to the effect of profile drag. However, when we calculate the propeller-induced velocity field in the wake the friction in the blade boundary layer and wake contraction may have significant effect on the axial and tangential components. In the present method the decrease of the axial component is taken as proportional to displacement thickness \( \delta^* \) of the boundary layer at the given blade section, which is estimated from the empirical identities. After that the corrections to the axial and tangential velocity components are derived from the condition of equal flowrates for the considered streamtube of radius \( r \) calculated with and without accounting for the velocity correction factor:

\[
\Delta \overline{W}_x^{vis} = -Z \frac{(\delta b / b_2)}{\pi (r / R) \sin \beta_{FVS}},
\]

\[
\Delta \overline{W}_\theta^{vis} = -\Delta \overline{W}_x^{vis} / \tan \beta_{FVS}.
\]

The effect of wake contraction can be significant at high propeller loading. In order to take it into account the simplest assumption about linear wake contraction is made and the correction factors are obtained from the momentum theorem for the contracted wake (Achkinadze et.al. 2003):

\[
\overline{W}_x^C (r_w) = \sqrt{\frac{1}{4} \left[ \frac{1 + \overline{W}_x (r_w)}{1 + \overline{W}_x (r_w)} \right] \frac{1}{K_w^2}},
\]

\[
\overline{W}_\theta^C (r_w) = \frac{\overline{W}_\theta [1 + \overline{W}_x (r_w)]}{[1 + \overline{W}_x^C] K_w^3},
\]

where \( r_w = r / K_w \) is the radius in the contracted jet that corresponds to the considered radius in the non-deformed jet and \( K_w \) is the wake contraction parameter, which is equal to the relation between the radii of the contracted and non-deformed jets \( K_w = R_w / R \).

The velocity field iteration procedure starts with calculation of the pod unit in uniform flow. After that the pod-induced velocity field is transferred into propeller design module where, added with prescribed hull wake in the propeller plane, it is used in the design calculation. The iterations between blade-row stages form the internal cycle of the iterative process. The mutually induced velocity fields due to propellers are calculated using the identities (3)-(10), after that the total velocity field induced by the both stages is used for recalculation of the pod unit. The described process repeated N times according to the user’s request. As calculations show usually 2 iterations between propellers (internal cycle) and 2 iterations between propellers and pod (external cycle) allow practical convergence.
3 PROPELLER ANALYSIS ALGORITHM

The current version of the analysis program allows the simulation of two-staged podded propulsor (exactly as in the design program) and the additional case of rudder behind propeller(s). The pod centerbody is considered as an axisymmetrical body of arbitrary shape, while strut(rudder) is represented as a wing of arbitrary geometry (including the case of twisted configuration). There is no restriction imposed on propeller blade geometry. Propeller and strut(rudder) are simulated as lifting bodies using the velocity based source boundary element method (BEM) while pod is treated as a displacing body. The program handles two main cases: steady and quasi-steady analysis. In the steady case the calculation is performed in the circumferentially averaged, radially and axially variable velocity field, while in the quasi-steady formulation the circumferentially variable inflow due to hull is prescribed in the propeller planes and the interaction between propellers and pod is considered in the spatially variable flow. The latest version of the analysis program allows calculation of controllable pitch propellers at off-design pitch setting. The main analysis outputs include propeller integral performance (in the unsteady case angular dependent blade forces are calculated), blade pressure distribution and cavitation domains estimated from the condition $-C_P \leq \sigma$, forces, pressure distributions and cavity domains on pod and strut(rudder) and total maneuvering forces on the unit. The velocity fields induced by the different components of propulsor and total velocity field around the unit are calculated.

3.1 Velocity based source BEM with modified trailing edge

This original BEM has been developed by the authors and earlier described in a number of publications (Achkinadze and Krasilnikov 2001), (Achkinadze et al. 2002), (Achkinadze and Krasilnikov 2003). The Figure 1 shows the principal representation of the lifting body by the hydrodynamic singularities used in this method. As one can see the sources of unknown strength $q$ are placed on the wetted surface and cavity surfaces in the case of cavitation in order to simulate the “thickness” effect. The doublets responsible for the “lifting” part of solution are placed on the mean surface inside the body and vortex wake behind it.

Figure 1: Representation of the lifting body in the authors’ BEM.

It is important to point out that in this kind of method the form of the vortex surface inside the body is not important as well as the form of distribution of the doublet strength on this surface. All that matters in the solution is the value of total circulation $\Gamma$ around given section of the body. Such a formulation allows obtaining of the integral equation from the main integral Green’s identity in the form of the Fredholm’s integral equation of the 2nd kind for non-cavitating flow

$$
\frac{q}{2} + \frac{1}{4\pi} \int_{S_\sigma} \frac{q}{R^3} \frac{\hat{R} \cdot \hat{n}}{R^3} ds + \frac{1}{4\pi} \int_{S_\sigma} \frac{\hat{\gamma}}{R^3} \frac{\hat{R} \times \hat{n}}{R^3} ds
$$

and in the form of the singular integral equation of the 1st kind (in the linearized form) for cavitating flow
\[ \frac{1}{4\pi} \int_{S_{CAV}} \left( q \cdot \frac{\vec{r} \cdot \vec{R}}{R^3} ds + \frac{1}{4\pi} \int_{\Gamma_0} \left( \frac{\vec{\gamma} \times \vec{R}}{R^3} \right) ds + \frac{1}{4\pi} \int_{\Gamma} \left( \vec{\Gamma} \cdot \frac{\vec{R} \times \vec{I}}{R^3} \right) ds \right) = \vec{V}_E \cdot \vec{n} - \vec{V}_\psi \cdot \vec{n}, \]

where \( q \) is the unknown source strength, \( \gamma \) is the vortex strength, \( \vec{n} \) is the surface normal, \( \vec{V}_E \) and \( \vec{V}_\psi \) are respectively transference and inflow velocities, \( \varphi \) is the absolute velocity potential, \( \sigma_Z \) is the local cavitation number. Thus, the important feature of the present method consists in that it allows mathematically correct formulation of the problem in the both cases of non-cavitating and cavitating flows due to the form of the obtained equations. It is different from the potential based BEM. However, the desirable form of the integral equations (11) and (12) is achieved when the vortex/(doublet) strength \( \gamma \) is known. This quantity (or more precisely, circulation \( \Gamma \)) is defined from the supplementary kinematic boundary condition, which is required on the so-called Modified Trailing Edge (MTE) – additional panel built as a continuation of the trailing edge (see Figure 1):

\[ \vec{V}_E \cdot \vec{n}_{MTE} = 0 \quad \text{on } S_{MTE}. \]

From the physical point of view this approach directly satisfies the Kutta-Joukowski condition on the realistic trailing edge since MTE along with (13) simulates the flow around reciprocal (return) point. This is another important difference of the present velocity based BEM from the potential BEMs where the Kutta-Joukowski condition is satisfied in iterations.

If the MTE is defined as a continuation of the mean surface

\[ \beta_{MTE}(r) = \arctg \left( \frac{\partial Y_\alpha}{\partial \xi} \right) \quad \text{at } \xi = \xi_{TE}, \]

then such the formulation meets the purely potential theory. However, MTE can be also used as a tool accounting for the viscous effects on circulation. The simple method consists in definition of the MTE bevel angle, which corresponds to the section lift coefficient in viscous flow, \( C_{L_V} \). The estimation of the \( C_{L_V} \) can be done using the empirical identities (as in the current program version) or direct viscous calculation of 2D profile equivalent to the considered blade section profile. The different conditions of the equivalence between 3D and 2D flows are discussed in (Achkinadze et.al. 2003). However, in the problem about search for the MTE bevel angle the simple condition of equal lift and moment coefficients of realistic 3D section and equivalent 2D profile can be employed. Along with corrections accounting for the profile drag this algorithm allows the improvement in prediction of propeller forces at off-design advance ratios \( J \) especially at very high and very low \( J \)-s.

In order to reduce the integral equations to the linear system the blade surface is represented by curvilinear quadrilateral boundary elements of constant strength \( q \). The blade surface is approximated using the mathematical tool of surface B-splines, and the singularities in behaviour of blade geometry elements at the blade tip and blade edges are taken into account (Achkinadze et.al. 2003). As it has been found during the code debugging, the curvilinear approximation of the blade surface appears to be quite efficient in the mentioned domains of strong surface gradient since it allows us to reduce the number of elements in comparison with flat or parabolic panel approximation. An example of computational grid on propeller blades can be seen on the Figure 2.

3.2 Steady, quasi-steady and unsteady formulations

The realistic flow field, in which the propeller operates, is spatially and time varying, and the inflow velocity vector has to be written as follows

\[ \vec{V}_\psi = \vec{V}_\psi(x_0, r_0, \theta_0, t), \]

where index “0” denotes the global fixed coordinate system, and \( t \) is the time. The circumferential non-homogeneity of the inflow is caused by a number of effects such as non-uniform hull wake in the site of propeller, non-zero heading angle of the pod drive (or shaft inclination) and interaction between propeller and
strut/rudder). In order to substitute this velocity field in the equations (11)-(12) we have to write it in the coordinate system fixed on propeller. The time parameter and the blade turn angle $\theta_p$ are connected through the identity

$$t = \frac{\theta_p}{\Omega},$$

where $\Omega$ is the propeller rate of revolution. Thus, in the propeller fixed coordinate system the inflow vector can be written as follows

$$\vec{v}_q = \vec{v}_q(x, r, \theta + \theta_p, \theta_p / \Omega),$$

and the unknown source and doublet strengths become time, or angular dependent. From the point of view of numerical implementation it means that the matrices of influence coefficient (left-hand parts of the integral equation) and inflow field (right-hand parts) change with blade turn angle $\theta_p$, and the solution of the integral equation should be performed in a step-wise manner at the given number of angular positions. The angular-dependent portion of the influence coefficients comes from the shed vorticity, which is a result of circumferential variation of $\Gamma$.

The steady solution is obtained at the assumption about circumferentially averaged flow. In this case the numerical solution of integral equation can be performed only for the key blade because of the flow symmetry. The blade pressure distribution is derived then from the Bernoulli equation. The first iteration of the unsteady solution assumes the invariable matrix of the influence coefficients (no shed vortices are taken into account) at the angular-dependent right-hand parts. In many cases (for example, when the inflow does not have very strong peaks) this so-called quasi-steady approach allows satisfactory precision as it has been confirmed by the test calculations.

The quasi-steady pressure distribution is defined from the Lagrange (unsteady Bernoulli) identity

$$p^* - (p_s + \rho gh) = -\rho \frac{\partial v^*_q}{\partial t} - \frac{\rho (v^*_s - V^*_s)}{2},$$

where the time derivative of the absolute velocity potential is calculated in the propeller-fixed coordinate system. The method of calculation of this derivative is described in (Achkinadze et.al. 2003).

3.3 Analysis of multi-component propulsor

Another important assumption of the developed quasi-steady approach concerns velocity fields, which are used when modeling the interaction between propulsor components. In principal, it is similar to one used in the design algorithm. The velocity fields induced by the blade-row stages are always considered as circumferentially averaged that allows the application of the fast and efficient computational scheme based on Hough and Ordway theory as described in Section 2.3. However, in the unsteady case all the other velocity fields (pre-set hull wake, fields induced by pod and strut) are spatially variable. In dependence upon available information the hull inflow can be specified only in the propeller plane or in a number of sections along the propulsor axis.

It has to be noted that in spite of that the “ideal” circulation obtained from the numerical solution appears to be changed due to effect of MTE bevel angle on section lift, the effect of the profile drag on circulation has to be taken into account in addition. This correction is obtained from the comparison of propeller torque coefficients calculated with ($K_{QV}$) and without ($K_Q$) accounting for the profile drag as follows:

$$\begin{cases}
\Gamma_V(r) = \Gamma(r)/C_{TV} \\
C_{TV} = K_Q / K_{QV}
\end{cases}$$

The system of correction factors accounting for the friction in the blade boundary layer and wake contraction effects remains exactly the same as in the design method.

If the propulsor component (strut, rudder, aft stage) is calculated in the propeller wake then the surface pressure distribution obtained directly from the Bernoulli (or Lagrange) identity without additional corrections appeared to be shifted regarding experimental values. A proper prediction of the pressure values in this case requires accounting for the “jump” of the Bernoulli constant, which takes place in the propeller disk due to
power delivered to the propeller. The simple correction factor applied in the present algorithm provides the absence of propeller-induced pressure in the infinitely far propeller wake that meets the well-known experimental fact. The change of the Bernoulli constant is estimated from this condition at one radius, which corresponds to the maximum axial propeller-induced velocity in order to obtain the maximum possible value of the pressure “jump”. After that the corrected pressure value on the body surface is obtained as follows

\[ C_p' = C_p - \Delta C_p^\infty, \quad \Delta C_p^\infty = 1 - \left(\frac{V_R^\infty}{V_E}\right)^2, \]

where \( C_p \) is the pressure coefficient defined from the Bernoulli identity without correction, \( V_R^\infty \) is the total relative velocity estimated in the far wake behind the propeller at the radius of maximum axial propeller-induced velocity. The satisfactory precision of this algorithm has been confirmed by the direct comparison between calculated and measured pressure distributions on the rudder surface behind operating propeller (Achkinadze et.al. 2003).

4 EXAMPLES OF NUMERICAL VALIDATION

The results of numerical validation of the design method have been published in (Achkinadze and Krasilnikov 1997), (Krasilnikov 1997), (Achkinadze et.al. 2000). The extensive numerical/experimental comparisons performed using the analysis program can be found in (Achkinadze et.al. 2001) (Achkinadze et.al. 2002), (Achkinadze et.al. 2003). Below we will consider the recent results obtained with analysis program applied to the different types of arrangements.

4.1 Open-water propeller performance and propeller induced velocity field

This example deals with a case of open single propeller and it has been mainly used for the verification of the velocity field algorithm since the data available from (Blaurock and Lammers 1988) contain the LDV measurements in the propeller wake. The general view of propeller with computational BEM grid can be seen on the Figure 2. The cosine grid of 11×20 elements (11 along the span, 20 along the chord) for one blade side has been used in this case. The Figure 3 shows the comparison between measured and calculated integral propeller performance. As it can be seen the satisfactory agreement in thrust and torque coefficients and propeller efficiency has been achieved in the whole range of \( J \) values. These results have been obtained using the algorithm of MTE bevel angle accounting for the viscous effects on circulation and corrections on section profile drag. On the Figures 4 and 5 the comparison between measured and predicted propeller-induced velocity fields is shown for the two sections downstream propeller: \( x/R = -0.6 \) and \( x/R = -2.0 \). In these calculations the full system of correction factors described above – reduction of circulation, friction on the blade surface and wake contraction (\( K_w = 0.92 \)) – has been applied, and the satisfactory agreement in terms of axial and tangential velocity components has been achieved.

4.2 Open-water performance of podded thrusters with contra-rotating propellers

The developed analysis program has been applied to simulation of two podded thrusters with pushing contra-rotating propellers (CRP) tested in the towing tank at MARINTEK. The main elements of blade-row stages are given in the Table 1. In the two studied cases the geometry of the pod and strut was kept identical while geometry of propellers was different as well as their location on the pod. The numerical/experimental comparison of the total integral performance of these arrangements is presented on the Figures 6 and 7. The total thrust coefficient was defined as a sum of thrusts of the forward and aft stages without accounting for the pod/strut drag and related to the diameter and rate of revolution of the forward propeller. Respectively, the torque coefficient was calculated through the total torque of propellers. A satisfactory agreement between theory and experiment has been achieved. In the first case good concordance is observed almost for the
Figure 2: General view and BEM grid of propeller BL2122.

Figure 3: Calculated and measured open-water performance of propeller BL2122.

Figure 4: Comparison between calculated and measured velocity components in the wake behind propeller BL2122. \( J=0.6 \). \( C_{Th}=1.30 \). Section \( x/R=0.6 \).

Figure 5: Comparison between calculated and measured velocity components in the wake behind propeller BL2122. \( J=0.6 \). \( C_{Th}=1.30 \). Section \( x/R=-2.0 \).

whole range of tested speeds. For the second arrangement discrepancy increases at low speeds and prediction of efficiency around design point is also worse because of the torque coefficient is underpredicted.
Table 1: Main elements of propellers used in the experimental/numerical study of pushing CRP thrusters at MARINTEK.

<table>
<thead>
<tr>
<th>Propellers</th>
<th>Case 1</th>
<th>Case 2</th>
</tr>
</thead>
<tbody>
<tr>
<td>Stage</td>
<td>FRW</td>
<td>AFT</td>
</tr>
<tr>
<td>Direction of rotation</td>
<td>Right</td>
<td>Left</td>
</tr>
<tr>
<td>Diameter, D [m]</td>
<td>0.19</td>
<td>0.167</td>
</tr>
<tr>
<td>Number of blades, Z</td>
<td>4</td>
<td>5</td>
</tr>
<tr>
<td>Blade area ratio</td>
<td>0.50</td>
<td>0.66</td>
</tr>
</tbody>
</table>

Figure 6: Comparison between measured and calculated total integral performance of CRP podded thruster. Case 1.

Figure 7: Comparison between measured and calculated total integral performance of CRP podded thruster. Case 2.

4.3 Prediction of maneuvering forces on rudder and podded drive

The experiments with propeller/rudder system and podded drive performed in the cavitation tunnel and towing tank at MARINTEK provided the authors with valuable information for the verification of the analysis program. The program of the tests for propeller/rudder system included the velocity measurement in the propeller wake, force measurements on the rudder behind the operating propeller and observation on cavitation on the rudder in the propeller wake. The measurements of maneuvering forces on the podded drive with single propeller embraced pulling and pushing arrangement in the range of heading angles from –45 to +45 degrees. The results of the measurements and program verification are discussed in details in (Achkinadze et.al. 2003). Below we will consider only the problem of prediction of maneuvering forces.

The general view of simulated propeller/rudder arrangement can be seen on the Figure 15. This figure also shows the calculated pressure distribution and cavitation domains on the rudder surface as it looks in the visual tool Pressv intended for the visualization of the mentioned characteristics. The main elements of propeller and rudder are summarized in the Table 2. The Figures 8 and 9 present the measured and calculated values of the rudder transverse force coefficient $K_T$ and rudder axial moment coefficient $M_{zax}$ versus rudder angle $\delta_r$ for
$J=0.8$ and $J=0.6$ respectively. A good agreement between theory and experiment can be noted for the transverse force almost in the whole range of rudder angles. The calculations also captured certain asymmetry in transverse force at positive and negative rudder angles observed in the experiment and caused by the direction of propeller rotation. However, the difference between calculated values of $K_Y$ at positive and negative rudder angles is smaller compared to the experimental figures. For example, at $J=0.6$ the experimental values were $K_Y=-0.191$ ($\delta_R=-15^\circ$) and $K_Y=0.221$ ($\delta_R=+15^\circ$) while calculation predicted $K_Y=-0.178$ ($\delta_R=-15^\circ$) and $K_Y=0.202$ ($\delta_R=+15^\circ$). The theory/experiment agreement for the rudder axial moment is noticeably worse especially for the higher rudder angles where the viscous effects presumably dominate. However, the absolute values of $M_{ZAX}$ are relatively small that makes difficulties not only for precise computation, but also for measuring of this quantity.

During the tests with single-propeller podded drive the axial and transverse forces and axial moment on the unit were measured at different heading angles for the pulling and pushing arrangements. The general view of the pulling arrangement with numerical grid is shown on the Figure 14. The change from pulling to pushing variant was done by the turn of the unit on 180 degrees around the strut axis and turn of propeller to keep the direction of its rotation, which in this case was right-handed. The results of the theory/experiment comparisons are given below for the two advance ratios close to 0.8 and 0.3 for the both variants. The assumption about fixed vortex wake spreading along the propulsor axis without deviation has been made throughout these calculations. As one can see from the Figures 10 and 11 a satisfactory agreement between measured and predicted values of

Table 2: Main elements of propeller and rudder used in the experimental/numerical study at MARINTEK.

<table>
<thead>
<tr>
<th>Prop diameter</th>
<th>D=0.25m</th>
</tr>
</thead>
<tbody>
<tr>
<td>Number of prop blades</td>
<td>Z=4</td>
</tr>
<tr>
<td>Blade area ratio</td>
<td>0.61</td>
</tr>
<tr>
<td>Pitch ratio at r/R=0.7</td>
<td>P/D=1.095</td>
</tr>
<tr>
<td>Rudder span/Prop diameter</td>
<td>L/D=1.251</td>
</tr>
<tr>
<td>Rudder aspect ratio</td>
<td>L/c=1.916</td>
</tr>
<tr>
<td>Distance between prop plane and rudder axis</td>
<td>X_R/D=0.5</td>
</tr>
<tr>
<td>Location of rudder axis, % from rudder LE</td>
<td>28.8</td>
</tr>
<tr>
<td>Vertical position of rudder regarding prop axis</td>
<td>Asymm.</td>
</tr>
<tr>
<td>Test conditions, J=0.8, 0.6, 0.4</td>
<td></td>
</tr>
<tr>
<td>Rudder angles, $\delta_R\in[-30^\circ;+30^\circ]$</td>
<td></td>
</tr>
</tbody>
</table>

Figure 8: Experimental and calculated values of rudder transverse force $K_Y$ and rudder axial moment $M_{ZAX}$. $J=0.8$.

Figure 9: Experimental and calculated values of rudder transverse force $K_Y$ and rudder axial moment $M_{ZAX}$. $J=0.6$. 
maneuvering forces has been obtained for the pulling arrangement in the range of heading angles from $-30^\circ$ to $+30^\circ$. The program has shown correct behaviour of the studied quantities with change of the heading angle and the values close to the experimental data in the mentioned range. The difference is slightly higher for the higher loading ($J=0.299$) and it becomes remarkable at heading angles about $\pm 45^\circ$. The discrepancy increases for the pushing variant (Figures 12 and 13) and the range of satisfactory agreement between theoretical and experimental predictions narrows to $\pm 15^\circ$. Presumably, the effect of deviation of propeller vortex wake plays here an important role. In the pulling case the pod centerbody and strut help the propeller wake to keep its orientation along the propeller axis (at least up to the certain heading angles) while in the pushing arrangement the helical vortices behind the propeller tend to deviate from their axial location influenced by the oblique flow already in the near propeller wake. The investigation of this effect using numerical model with oblique vortex wake is considered as an important future task.

Figure 10: Measured and calculated maneuvering forces on pulling unit. $J=0.799$.

Figure 11: Measured and calculated maneuvering forces on pulling unit. $J=0.299$.

Figure 12: Measured and calculated maneuvering forces on pushing unit. $J=0.813$.

Figure 13: Measured and calculated maneuvering forces on pushing unit. $J=0.3088$.

5 PROGRAM SHELL AND VISUAL TOOLS

The developed design/analysis system is the standard Windows application suitable to use under Windows 98/Me, Windows NT4/2000/XP platforms. The design and analysis programs have separate shells, but they are connected via easy data import/export functions. For example, as soon as the design calculation is completed the resultant geometry can be transferred directly into the analysis program by pushing only one button.

The common part of both programs is the special tool Propeller Viewer intended for visual control and manual correction of propulsor geometry. An example of Propeller Viewer window showing the podded thruster
considered in the section 4.3 with numerical grid is given on the Figure 14. This window consists of the following three windows: 3D image of propulsor, table window (where the propeller blade geometry elements are tabulated) and plot window (where the same distributions are given as plots versus radial coordinate). Both table and plot windows are available for the manual edition, and as soon as the changes are done they are reflected on the 3D image. All the other elements of propulsor are available for the edition either through the Propeller Viewer menu functions or through the design/analysis input data dialogs. Besides the general view of propulsor the Propeller Viewer options allow the viewing of the blade sections by the arbitrary plane, computation of the cartesian coordinates of the blade surface and estimation of the gaussian surface curvature, which helps to find possible “cup” domains on the blade surface.

The pressure distributions on propeller blades, pod and rudder(strut) obtained as a result of the analysis are available as 3D and 2D colour diagrams in the visual tools Pressv (3D) and Pressf (2D). Both programs allow the representation of the surface pressure distribution in different dimensionless forms: pressure coefficient, pressure coefficient related to the local cavitation number, cavitation margins in per cent and, finally, cavitation domains estimated from the “fully wetted” condition – $C_p \leq \sigma$. An example of 3D pressure distribution on the rudder surface visualized in the Pressv program is shown on the Figure 15. This case meets the arrangement considered in the section 4.3. However, the cavitation number corresponds to the cavitation tests and the cavity domain near the rudder leading edge can be seen. The 2D window of Pressf program also shows the chordwise pressure distributions as plots.

All these tools together allow the user to organize the design process as a dialog with checking of geometry and hydrodynamic characteristics at each step and making proper corrections if it is necessary. After the design/analysis procedure is finished the propeller blade geometry is available for drawing and 3D description for milling machines. The Figure 16 shows an example of simple blade drawing and 3D blade surface model in AutoCAD generated by the Drawing and 3DXYZ tools included in the current program kit. In addition, the visual tools of the 3DXYZ program allow manual edition of the blade section geometry including the fine details of the leading and trailing edges in the special windows shown on the Figure 17.

Figure 14: Main window of Propeller Viewer showing the general view and computational grid of podded thruster.
Figure 15: Pressure distribution and cavitation domains on the rudder surface visualized in the Pressv program. $J=0.6$. Starboard. $n=15\text{Hz}; \sigma^{(P)}=6.677; \delta_R=20^\circ$. Comparison with experiment.

Figure 16: Propeller blade drawing and 3D blade surface model in AutoCAD generated by the visual program tools.

6 EXAMPLES OF USE OF DESIGN/ANALYSIS PROGRAM IN PRACTICAL DESIGNING

6.1 Pod effect on pitch and camber of blade-row stage

This example shows the importance of accounting for the pod body in the design calculation. It also illustrates the fact that use of stock propeller designed for the “open water” conditions on the pod may lead to a noticeable difference between obtained and expected performance. The pod configuration used in this case was similar to one shown on the Figure 14. The pulling variant was considered. The main elements of designed
propeller were as follows: number of blades \( Z = 4 \), blade area ratio \( 0.48 \), diameter \( D = 0.21\text{m} \). The standard NACA,\( a=0.8/66\mod \) sections were used. The design point was: \( n = 12\text{Hz}, \ J = 0.7, \ K_Q = 0.0245 \). In the first column of the Table 3 the lifting-line force prediction from the design program is given and the pitch and camber distributions along the radius are shown on the Figures 18 and 19. The results of the BEM analysis of propeller have shown that design geometry provides the expected integral performance (the difference in \( K_Q \) amounted only +1.6%). The analysis of the same geometry on the pod body (third column in the Table 3) revealed the increase of propeller forces (\( K_Q \) increased on 6.0%, \( K_T \) increased on 6.7%). After accounting for the pod body in the design calculation (2 iterations between propeller and pod) the new geometry was obtained and, as it can be seen from the Figure 19, it is characterized by decrease of the pitch values, especially remarkable at the root sections. The analysis of the new propeller has shown the integral performance to be closer to expected values (+3.2% difference in \( K_Q \), +3.4% in \( K_T \)).

Then, the cavitation analysis of propeller on pod has been performed and the obtained cavitation bucket is presented on the Figure 20. The left branch corresponds to the inception of cavitation near the leading edge on the suction side, the bottom of the diagram corresponds to the midchord cavitation on the suction side, and the right branch indicates the cavitation near the leading edge on the pressure side. As one can see, the design point \( J = 0.7 \) appeared to be closer to the left branch of the cavitation diagram, which – along with the slightly higher propeller forces – is in line with the results obtained earlier by the authors from the analysis of the experimental data with designed propellers (Achkinadze and Krasilnikov 2000). The improvement of design geometry consisted in decrease of the blade pitch at simultaneous increase of the section camber by the iterative implementation of the analysis code. The new pitch and camber distributions are compared with the previous two cases on the Figures 18 and 19. The propeller forces are given in the last column of the Table 3, and the cavitation diagram for the new geometry is shown on the Figure 20.

Table 3: Computation of propeller forces for open water and podded arrangements.

<table>
<thead>
<tr>
<th>Forces</th>
<th>Design open-water</th>
<th>Analysis open-water</th>
<th>Analysis on pod</th>
<th>Design N1 on pod</th>
<th>Analysis N1 on pod</th>
<th>Analysis N2 on pod</th>
</tr>
</thead>
<tbody>
<tr>
<td>( K_Q )</td>
<td>0.0245</td>
<td>0.0249</td>
<td>0.0264</td>
<td>0.0245</td>
<td>0.0253</td>
<td>0.0246</td>
</tr>
<tr>
<td>( K_T )</td>
<td>0.1466</td>
<td>0.1501</td>
<td>0.1600</td>
<td>0.1483</td>
<td>0.1534</td>
<td>0.1502</td>
</tr>
<tr>
<td>( \eta_D )</td>
<td>0.6666</td>
<td>0.6696</td>
<td>0.6747</td>
<td>0.6744</td>
<td>0.6758</td>
<td>0.6788</td>
</tr>
</tbody>
</table>
The better agreement between required and predicted propeller forces is obvious. Also the new cavitation bucket is wider compared to the first design, and the design point is located closer to the middle of the bucket bottom that is always one of the goals of successful blade design. The following improvement in cavitation behaviour requires modification of the section mean line and thickness distributions.

6.2 Modification of the leading edge: effect on pressure side cavitation on highly skewed propellers.

During the tests with highly skewed controllable pitch propeller P1302 performed in the cavitation tunnel at MARINTEK the intensive pressure side cavitation was observed at off-design low-pitch setting. In order to avoid this negative phenomenon a special modification of the leading edge form was done. It consisted in lifting of the leading edge at the same maximum camber and design pitch distributions and slightly reduced chord length of the outer sections. After that the amount of pressure side cavitation was significantly reduced. The analysis program was applied to investigation of possible reasons of these effects. On the Figure 21 the calculated pressure distributions on the pressure side of the initial P1302 and modified P1302mod propeller designs are shown. As one can see the initial design, indeed, reveals the pressure side cavitation and, besides, the strong oscillations of the pressure are visible. In authors’ opinion, it was caused by the excessive camber of the blade sections and might lead to the possible flow separation in these domains and, as a consequence, to the intensive cavitation of vortex type. After the modification the pressure distribution on the pressure side appeared to be much smoother, with the only pressure peak near the leading edge, and the cavitation was significantly

Figure 18: Design pitch distributions for podded propeller.

Figure 19: Design camber distribution for podded propeller.

Figure 20: Calculated cavitation buckets for the design of podded propeller.
Figure 21: Pressure distributions and cavitation domains on pressure side of propellers P1302 and P1302mod. (Design pitch $P/D(0.7)=1.50$, actual pitch $P/D(0.7)=1.100$, $J_r=1.070$, $\sigma_{rs}^{V_r}=0.211$).

Figure 22: Effect of lifted leading edge on pressure distribution along the blade sections of highly skewed propeller (plots generated in the Pressf visual program).

reduced. The only narrow strip of cavitation predicted by the program was also observed in the experiment. The initial and modified blade sections are shown on the Figure 22 along with the chordwise pressure distribution.
7 CONCLUSIONS

The developed interactive program system allows design and analysis calculations of marine propulsors of different types including single and contra-rotating shaft arrangements, propeller(s)/rudder systems and two-staged podded arrangements. The visual tools included in the program kit provide the user with the wide possibilities of input/output data control, manual correction of principal arrangement and blade geometry, edition of the blade sections, drawing and creation of the 3D blade model for milling machines.

The developed design and analysis programs have passed through extensive numerical/experimental verifications. The results presented in this paper show a good agreement between theory and experiment in terms of integral performance for single open propellers and CRP podded thruster, and propeller-induced velocity field. The numerical simulation of a rudder behind operating propeller and single-screw pulling/pushing unit has been performed. Satisfactory agreement with the measurement has been obtained for the transverse force component. The problem of prediction of the axial moment on rudder and podded drive at high heading angles requires additional attention and, probably, more profound accounting for viscous effects.

Current authors’ efforts are addressed to full calculation of cavity patterns on propeller blades and rudder and investigation of the effect of propeller wake deviation on propulsor performance. The inclusion of the duct/tube analysis in the described program system is seen as an important future task.

REFERENCES


